v1.2

for Windows™

TimingDesigner™

the interactive timing analyzer

User Manual

CHRONOLOGY

v1.2

for Windows™

TimingDesigner™

the interactive timing analyzer

User Manual

CHRONOLOGY

Information in this document is subject to change without notice and does not represent a commitment on the part of Chronology Corporation. The software described in this document is furnished under a license agreement or non-disclosure statement. The software may not be used or copied only in accordance with the terms of the agreement. It is against the law to copy the software on any medium except as specifically allowed in the license or non-disclosure agreement. No part of this manual may be reproduced or transmitted in any form or by any means, electronic or mechanical, including photocopying and recording, for any purpose without the written permission of Chronology Corporation.

© 1991 Chronology Corporation. All rights reserved.

TimingDesigner is a trademark of Chronology Corporation.
Microsoft is a registered trademark of Microsoft Corporation.
Microsoft Windows is a trademark of Microsoft Corporation

Table of Contents

1: Welcome!

How To Proceed 1-2
For Further Assistance 1-2

2: Installation

Before You Begin 2-1
Checklist 2-1
Installing TimingDesigner 2-3
Installing the TimingDesigner Files 2-3
Installing the Authorization Key 2-5

3: Tutorial

Before You Begin 3-1
Starting TimingDesigner 3-2
Using TimingDesigner 3-4
Exploring TimingDesigner 3-5
Create a signal waveform 3-6
Create another signal waveform 3-7
Add a delay 3-8
Establish delay parameters 3-9
Add an annotation 3-10
Examine attachments 3-11
Add a clock 3-12
Move the clock 3-13
Add another delay 3-14
Add a constraint 3-15
Analyze and correct the constraint violation 3-16

Add a variable 3-17
Exit TimingDesigner 3-19
Before Moving On... 3-20

4: About TimingDesigner

Working with TimingDesigner 4-2
Timing Diagrams 4-3
Waveforms 4-4
Edges and States 4-5
Edges 4-5
Uncertainty 4-5
Edge Display 4-6
States 4-6
Delays 4-7
Constraints 4-9
Margins 4-9
Numerical Values 4-10
Terminology 4-11

5: TimingDesigner Basics

Starting and Exiting TimingDesigner 5-2
TimingDesigner Windows 5-2
The Diagram Window 5-2
Spreadsheets 5-3
The Parameter Spreadsheet 5-3
Library Spreadsheet 5-4
Working With Files 5-5
Opening a file 5-5
Clearing a file from the screen 5-6
Saving your work 5-6
Controlling TimingDesigner Windows 5-7
Selecting a window 5-7
Arranging windows 5-8
The Tool Bar and Tools 5-8
The New waveform buttons 5-9
The Tool buttons 5-9

The State buttons 5-9
The Delete button 5-9
Tools: A closer look 5-10
Editing Text 5-12
Working with Objects 5-12
Selecting:How and what 5-12
Selecting overlapping objects 5-13
Selecting multiple graphic objects 5-14
Selecting waveforms or spreadsheet rows 5-14
Working in the Spreadsheets 5-14
Spreadsheet columns 5-14
Controlling spreadsheet column widths 5-15
Editing in the spreadsheets 5-16
Inserting a spreadsheet row 5-16
Deleting a spreadsheet row 5-17

6: Waveforms

Signals 6-1
Setting initial state 6-1
About toggle patterns for new waveforms 6-2
Adding signals 6-2
Deleting signals 6-3
Naming signals 6-3
Moving signals 6-4
Clocks 6-5
Adding a clock 6-5
Moving clocks 6-5
Inserting and deleting clock cycles 6-5
Setting and Changing Clock Attributes 6-7
Clock Attributes dialog box fields 6-7
Using Formulas in Clocks 6-9
Creating Derived Clocks 6-10
Derived clock example 6-10

Locking and Unlocking Waveforms 6-12

7: Edges and States

Working with Edges 7-2
Using sloped or unsloped edges 7-2
About toggle patterns 7-3
Adding edges 7-3
Deleting edges 7-4
Moving edges 7-4
Changing edge state 7-5
Setting and Changing Edge Attributes 7-6
Edge Attributes dialog box fields 7-6
Locking Edges 7-7
Using Snap to Place Edges 7-7
States 7-8
Deleting states 7-8
Changing states 7-8

8: Delays

About Delays 8-2
Delay formulas 8-3
Working with Delays 8-3
Adding a delay 8-3
Deleting a delay 8-4
Entering numeric values for a delay 8-4
Hiding a delay 8-4
Showing a hidden delay 8-5
Setting and Changing Delays 8-6
Delay Attributes dialog box fields 8-6
Setting and Changing Display Preferences 8-7
Setting and changing global defaults 8-7
Display Preferences dialog box fields 8-8

Labels 8-8
Hidden options 8-9
Sloped edges option 8-9
Show uncertainties option 8-9
Multiple Delays 8-10
Target Types 8-10
Critical paths and the combination rule 8-11
About target edge type first 8-12
About target edge type last 8-13
About target edge type outside 8-14

9: Constraints

How Constraints Work 9-2
About Measure 9-2
Constraint Values 9-3
Margins 9-3
Calculating margins 9-3
About Signal Skews 9-4
Working with Constraints 9-4
Adding a constraint 9-4
Deleting a constraint 9-4
Entering numeric values for a constraint 9-5
Hiding a constraint 9-5
Showing a hidden constraint 9-6
Setting and Changing Constraints 9-7
Constraint Attributes dialog box fields 9-7
Labels 9-8
Working with Signal Skews 9-8
Skew example 9-8

10: Variables, Libraries, and Formulas

Variables 10-2
Variable names 10-2
Creating a variable 10-2
Deleting a variable 10-3
Moving a variable 10-3
Using a variable 10-3

Using Libraries 10-4
Using library variables (importing) 10-4
Sending variables 10-4
Importing by reference 10-5
Updating variables 10-5
Library File Format 10-5
Omitting fields 10-6
Commas in fields 10-6
Managing Library hiles 10-6
Formulas 10-7
Formula elements 10-7
Mathematical operations 10-8
Operations in formulas 10-8
Operation order 10-8
Functions 10-9
About Nulls 10-9

11: Text Annotations

Working with Annotations 11-2
Adding annotations 11-2
Editing annotations 11-3
Moving annotations 11-3
Deleting annotations 11-3
Attaching Annotations 11-4
About SmartSnap 11-4
Setting the Annotation Font 11-5
Annotation Font dialog box fields 11-5

12: Printing and Exporting

Printing 12-1
Printing timing diagrams 12-1
About the Print dialog box (Diagram Window) 12-2
What to print 12-2
How to print 12-2
Commands and options 12-2
Printing spreadsheets 12-3
About the Print dialog box (spreadsheet windows) 12-3
Choosing a printer 12-3

Exporting Graphics 12-4
Exporting timing diagrams 12-4
Copy to Clipboard dialog box (Diagram Window) fields 12-4
Exporting spreadsheets 12-5
Copy to Clipboard dialog box (spreadsheets) fields 12-5

13: Other Features and Functions

Changing Sources and Targets 13-2
Taking Time Measurements 13-3
Working with the vertical reference cursor 13-4
Working with the marker line 13-4
Setting the marker line location 13-4
Moving and removing the marker line 13-4
Horizontal Reference Lines 13-5
Setting the Diagram and Spreadsheet Font 13-6
Saving TimingDesigner Options 13-8
TD.INI File Format 13-8
Merging Timing Diagrams 13-10
Zoom 13-11
Grids 13-12
Grid dialog box fields 13-13
Jumping to a Diagram Location 13-14

1

Welcome!

Take a moment to look over the organization of this manual. It will help you quickly locate the information you need.

Chapter 1	**Welcome!** orients you to the documentation.
Chapter 2	**Installation** tells you how to install TimingDesigner on your system, and how to start and end TimingDesigner sessions.
Chapter 3	**Tutorial** gives you hands-on experience with TimingDesigner. The steps of the tutorial cover the complete process of creating a timing diagram.
Chapter 4	**About TimingDesigner** introduces terms and concepts you need to know to understand the product, and gives a framework for using TimingDesigner.
Chapter 5	**TimingDesigner Basics** details the basics of working with TimingDesigner, focusing upon functions and features that are applicable throughout the product.
Chapter 6	**Waveforms** discusses signal and clock waveforms, how they differ, and how to work with them.
Chapter 7	**Edges and States** explains how to actually draw waveforms by working with edges, and how to work work with states.
Chapter 8	**Delays** explains how delays work, and then covers how to add, edit, and use delays. The chapter ends with a discussion of multiple source delays.
Chapter 9	**Constraints** explains how constraints work, and then covers how to add, edit, and use constraints. The chapter includes information on margin, measure, and signal skews.

Chapter 10	**Variables, Libraries, and Formulas** discusses a number of important topics that focus on the way numerical values are used in TimingDesigner.
Chapter 11	**Text Annotations** explains how to use text annotations and how to work with text attachments.
Chapter 12	**Printing and Exporting** discusses printing and using the Clipboard to export data to word processing and page layout applications.
Chapter 13	**Other Features and Functions** covers a number of diverse TimingDesigner operations. Scan through this chapter to make sure that you are using all of the powerful features of TimingDesigner.

How To Proceed

1. Complete your registration form and drop it in the mail now! Doing so will ensure that you receive updates.
2. Install TimingDesigner according to the steps in Chapter 2, "Installation."
3. Work through the tutorial in Chapter 3, "Tutorial."
4. Browse through Chapter 4, "About TimingDesigner," to understand the terms we use and the major concepts behind TimingDesigner.
5. Begin using TimingDesigner. If you are familiar with Windows applications, the process will come easily to you. Keep the manual open to Chapter 5, "TimingDesigner Basics," which summarizes the major interface components of the product.

For Further Assistance

If you need further assistance, please call Chronology Technical Support. Hours are weekdays from 8:00 a.m. to 5:00 p.m., Pacific Time, and the numbers are listed on the back cover of this manual.

Before you call, take the time to clarify the problem you are having:

- Make sure that the problem is repeatable.
- Consult the manual carefully to see if there is help available.

- Try to determine if the problem is related to the Microsoft Windows software. If you're experiencing printing problems, for example, see if you can print using other applications, such as Write and Paint.

Please have the following information available:

- Hardware configuration—memory capacity, processor type and speed, and so forth.
- Version number—obtained by selecting the About TimingDesigner command from the Help menu in the Diagram Window.
- Your serial number—listed inside the manual, on your TimingDesigner disk, and on the authorization key.

2

Installation

Before You Begin

To make sure that the installation goes smoothly, and to ensure that your system is capable of running TimingDesigner, perform the following checks. If you cannot successfully perform any of the checks, there is a problem with your Microsoft Windows installation that must be resolved before you can work with TimingDesigner.

Checklist

✔ **Do you have the necessary hardware?**

To install and use TimingDesigner, you need the following hardware:

- PC using a 286/386/486 processor
- EGA, VGA, or higher resolution monitor
- One hard drive, and one 1.2MB or one 1.44MB floppy drive
- 2MB RAM
- Mouse

✔ **Is Microsoft Windows version 3.0 installed?**

You must have Windows to run TimingDesigner.

✔ Do you have enough memory?

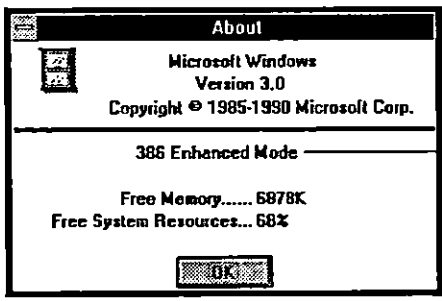
Although Windows will run in Standard Mode with 1MB of memory, you must have a total of at least 2MB of memory to run TimingDesigner. Your computer's documentation will tell you how much memory you have.

✔ Can you print from Windows?

Make sure that your printer is working properly with Windows. To check this, start Write or Notepad and print out a simple document (such as your AUTOEXEC.BAT file) on the same printer or plotter that you plan to use with TimingDesigner.

✔ Can you run Windows in Standard or Enhanced mode?

To determine this, start Windows, and then select the About Program Manager... command from the Help menu in the Program Manager. The resulting dialog box should clearly state that you are either in Standard Mode or Enhanced Mode. If the dialog box states that you are in Real Mode, TimingDesigner will not run.



This area of the About Program Manager dialog box tells you the Windows mode you are in.

Installing TimingDesigner

The steps to follow for installing TimingDesigner are given below. Complete installation consists of installing the TimingDesigner program files, creating a Windows Program Item, and installing the authorization key.

Installing the TimingDesigner Files

The installation process will install an executable file named TD.EXE in whichever directory you designate. The default installation directory is C:\TD. If the directory you specify does not exist, the installation procedure will create it auto- matically. The installation process will also create a subdirectory named SAMPLE in the specified directory. Sample diagram and library files supplied by Chronology are installed in this directory.

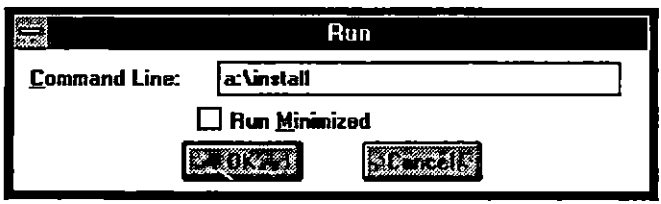
Once the installation process copies the files, a group window titled CHRONOLOGY will be created and an icon with the description TimingDesigner will be installed in CHRONOLOGY window.

✔ To Install the TimingDesigner files

1. Start Windows
2. Place the installation diskette in the proper drive.

   Note: The rest of these instructions will assume that the drive containing the TimingDesigner diskette is A:, and that you are going to install TimingDesigner on C:. Substitute drive letters as appropriate to your environment.

3. Select the Run command from the File menu of the Program Manager or the File Manager, whichever you prefer to use. If you install from File Manager, you must make the diskette drive the default drive.



4. In the Command Line field, enter a:\install. The installation procedure begins.
5. When you are asked about the installation directory, you can choose the default installation directory, C:\TD, or enter a new drive:path specification. When you are satisfied with the path, select OK, and the installation process continues. A message stating the file currently being transferred, and its destination, is shown. A Percent Complete indicator is updated for each file.
6. Once all files have been copied to your hard drive, a new group window will appear titled CHRONOLOGY and an icon labeled TimingDesigner will appear. Finally, a dialog box stating that the installation procedure is complete will appear. Press Ok to activate the Program/File Manager.

If you choose to place the TimingDesigner icon in a different group window, you may use the standard Windows procedure for moving icons between groups. You should also then delete the CHRONOLOGY group window if you have no other purpose for it.

Installing the Authorization Key

TimingDesigner uses a hardware-based system to authorize your computer to run the software. The system consists of a small "key" that installs on your parallel printer port. The key should not cause any problems, with either your computer or your printer.

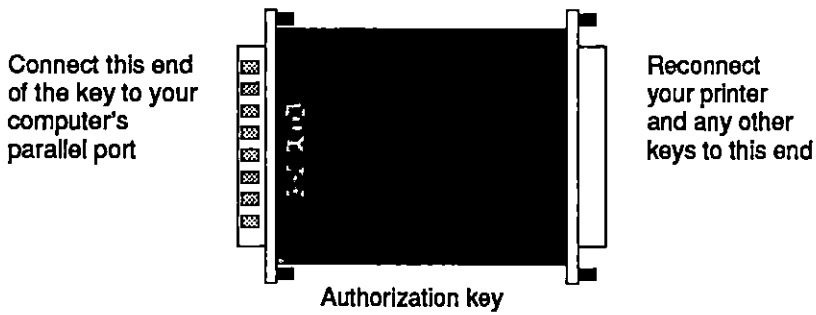
✔ To install the authorization key

1. Locate the parallel port at the rear of your computer. If you are unsure as to which connector is the parallel port, refer to your computer system documentation. (PC parallel ports use a 25-pin female connector.) If a printer cable or other authorization keys are attached to the parallel port, disconnect them temporarily.

   Note: If you have more than one parallel port, you can place the key on any port connector.

2. Locate the key that was shipped with your TimingDesigner software. It looks like a printer connector, and has a male and female end (see diagram below).

3. Connect the male connector of the key to your computer's parallel port. (Notice that the word "computer" and arrows indicate which end connects to the computer.) Screws on the key allow you to fasten it to your system.



Authorization key

4. If you disconnected a printer or other authorization keys from the parallel port, reconnect them to the female end of the key.

3
Tutorial

This tutorial introduces you to some of the major features of TimingDesigner. By perfoming the actions step-by-step, you'll gain valuable insight into how TimingDesigner works.

The tutorial provides a structured introduction to TimingDesigner—it's not an exhaustive look at all of TimingDesigner's capabilities. Once you've completed the tuorial, you can begin working with TimingDesigner on your own designs, referring to the later chapters in this manual for more information on other features and functions.

Before You Begin

Do the following to ensure that the tutorial runs smoothly.

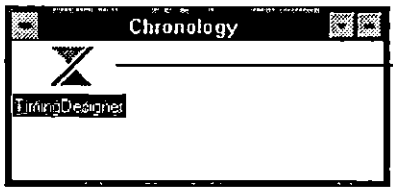
If TimingDesigner has been in use on your system, a settings file named TD.INI may exist. This file keeps track of certain TimingDesigner options. If it exists, your screens may not match those in this tutorial. Exit TimingDesigner, if necessary, and do one of the following:

- Delete the file. It is located in the \WINDOWS directory.
- Rename the file to TDINI.BAK. This will allow you to restore the file when you are through working the tutorial.

Starting TimingDesigner

✔ To start TimingDesigner

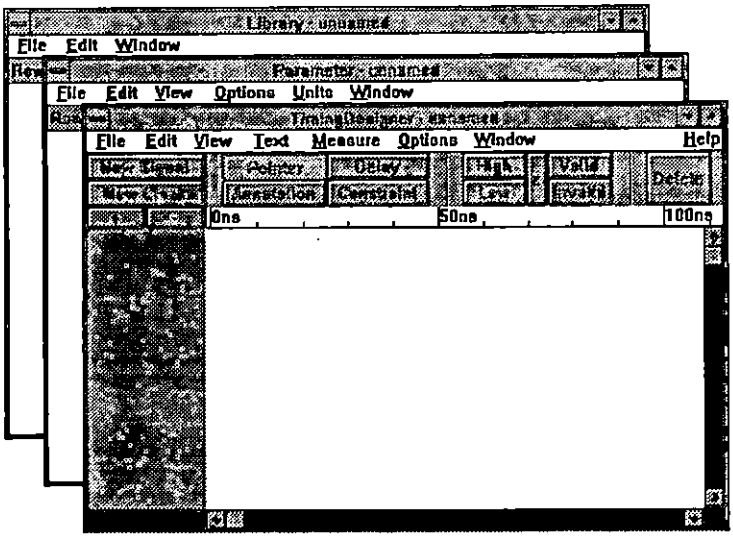
1. Start Windows, if necessary.
2. Double-click on the TimingDesigner icon. If you followed the default installation procedure, the icon is located in the Chronology group in the Windows Program Manager.



When TimingDesigner starts, three windows appear (shown on the following page):

- The Diagram Window is the active window (the one in the front). Timing diagrams are drawn here.
- The Parameter Spreadhsheet is behind the Diagram Window. The Parameter Spreadsheet consists of rows that contain the values of delays and constraints, and definitions of variables that are used in formulas.
- The Library Spreadsheet is the window at the back. The contents of a Library Spreadsheet are similar to an on-line databook and contains variables used in many designs.

3. Select the Tile (no library) command from the Window menu in the Diagram Window. This command tiles the Diagram Window and the Parameter Spreadsheet, with the Diagram Window on the bottom taking up about two-thirds of the screen.



When TimingDesigner starts, all three windows are "cascaded."

Using TimingDesigner

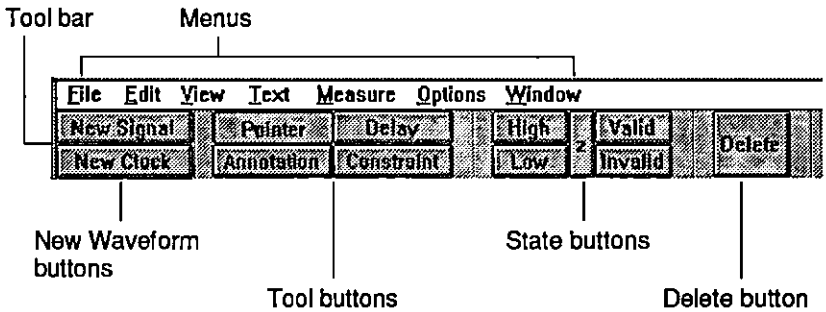
Most TimingDesigner activities are simple and intuitive. For example, waveforms are drawn by clicking with the mouse in the waveform area. In addition to the mouse, the program uses menus and a tool bar.

Note: Unless indicated otherwise, use the left mouse button when directed to "Click."

The menus give you access to important commands, such as saving files and exiting the program.

The tool bar gives you instant access to commands and functions through the use of buttons. When steps in the tutorial refer to buttons, you'll find them in the tool bar. The buttons perform these functions:

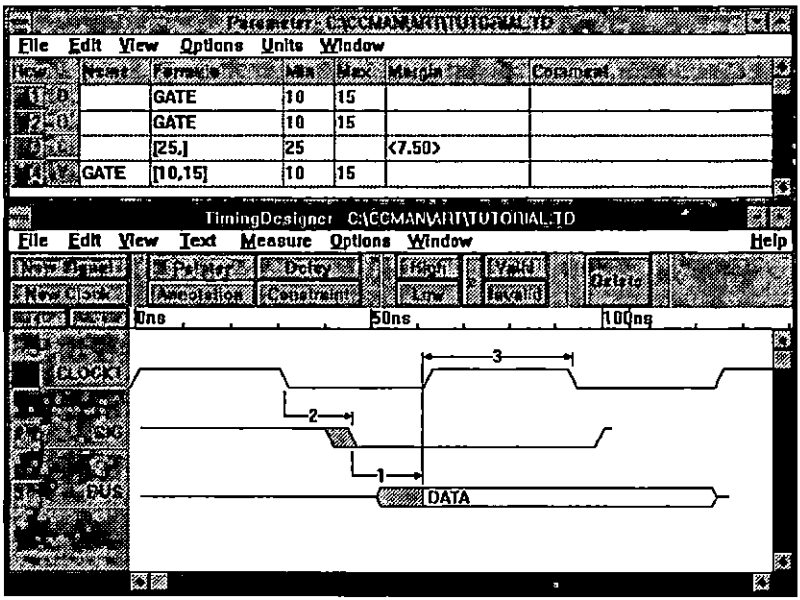
- The New Waveform buttons adds new signal or clock waveforms to the design.
- The Tool buttons select various mouse-based pointers that perform important functions, such as adding text or delays.
- The State buttons set the state of states and edges.
- The Delete button deletes objects, such as states, edges, and delays.



The menus and tool bar give you access to TimingDesigner's commands.

Exploring TimingDesigner

The following exercises introduce you to the power of TimingDesigner. By following working step-by-step, you'll be able to create and analyze the timing diagram below.

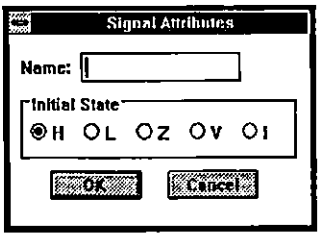


This is how the timing diagram and Parameter Spreadsheet will look when you complete the tutorial.

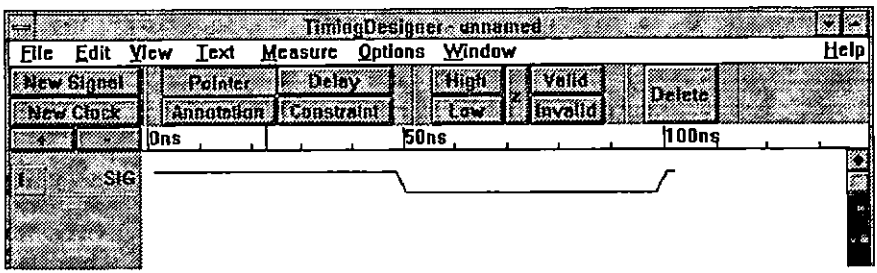
Create a signal waveform

Usually, the first thing done in timing diagram is to add a signal waveform, and then draw the waveform's states and edges.

1. Click the New Signal button. The Signal Attributes dialog box appears.



2. In the Name field, type **SIG**. Notice that the Initial State for this waveform is set to "H" (high). Click OK to accept this signal.
3. The waveform SIG appears just below the ruler. The name you specified in the step above is on the right. To the left of the name is a field with a 1 in it. This is the system-generated number for this signal. You cannot edit this field.
4. To add the first edge, move the Pointer tool into the waveform area. Using the hairline indicator in the ruler as a guide, position the Pointer tool at 50ns, at the level of the waveform name. Click the left mouse button. A High state with a Low edge appears.
5. To add a second edge, move the Pointer tool to the right, and click at 100ns. A Low state with a High edge appears.



This is how the timing diagram looks when you complete the steps above. The edges for SIG are at approximately 50ns and 100ns.

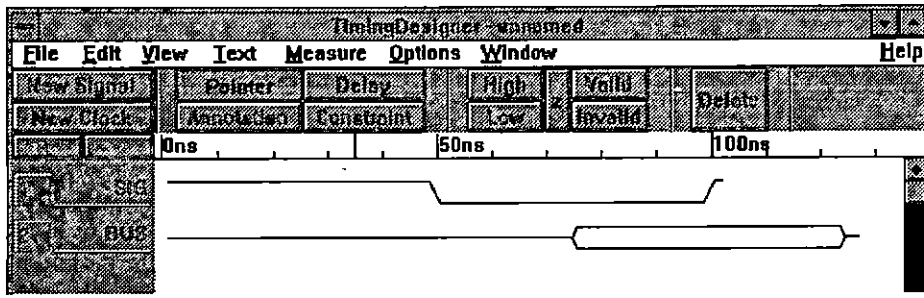
Hint: Most edges in your design only need to be placed approximately because, through the use of delays, TimingDesigner will actually position the edges for you.

Create another signal waveform

The waveform created in this exercise is a data bus, and it uses a different set of states, Z and Valid.

1. Click the New Signal button. The Signal Attributes dialog box appears.
2. Type **BUS** in the Name field.
3. To set the initial state of this waveform, click the Z option in the Initial State field, and then click OK.
4. Add an edge by clicking at 75ns. A Z state with a Valid edge appears. Click at 125ns for the next edge. A Valid state with a Z edge appears.

Hint: When you change the default first state of a waveform to Z, the waveform states automatically toggle between Z and Valid with each new edge.

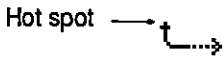


This is how the timing diagram looks when you complete the steps above. The edges for BUS are at approximately 75ns and 125ns.

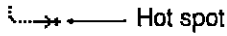
Add a delay

A delay represents the propagation delay between an input signal and an output signal, and implies a causal timing relationship between two edges. Once a delay is set, the software automatically maintains the time relationship between the two edges.

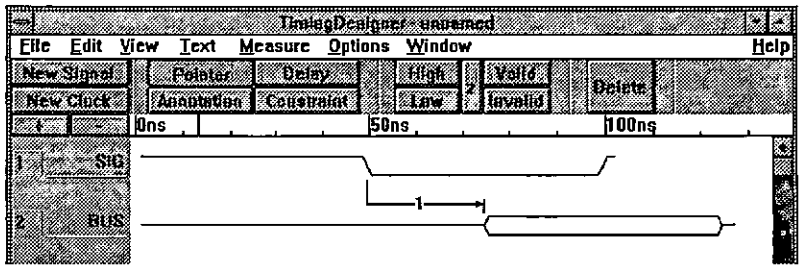
1. Click the Delay button. The Delay Source tool appears. Its hot spot looks like a "+" in the solid left half.



2. Set the source edge of the delay by clicking with the hot spot on the first falling edge of SIG. The Delay Target tool appears. Its hot spot looks like a "+" at the end of the arrow head.



3. Set the target edge of the delay by clicking with the hot spot on the first edge in BUS.



Establish delay parameters

In the last exercise, you created a delay by indicating graphically the involved edges. Notice that the system automatically entered a row in the Parameter Spreadsheet for the delay. The Row column now contains a row, 1, and the row type, D (delay).

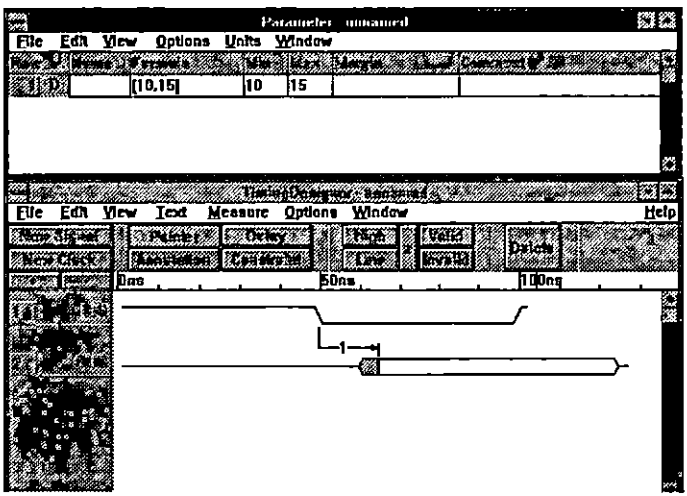
In this exercise, you'll specify the numeric relationship between the edges of the delay by entering data into the Parameter Spreadsheet.

1. Establish the minimum for the delay.

   In the Parameter Spreadsheet, click in the Min cell, and then type **10.** Press Tab. The value is simultaneously copied to the Max column and to the Formula cell. In the Diagram Window, notice that the target edge of the delay on the waveform BUS moves to the left—to 10ns from the source edge (it was about 25ns from the source edge).

2. Establish the maximum for the delay.

   Select the 10 in the Max cell (if it is already highlighted it is selected), and then type **15.** Press Enter. In the Diagram Window, an uncertainty (a gray area) appears. This uncertainty indicates the earliest and latest time that the edge can occur, based on the Min and Max values of the delay.

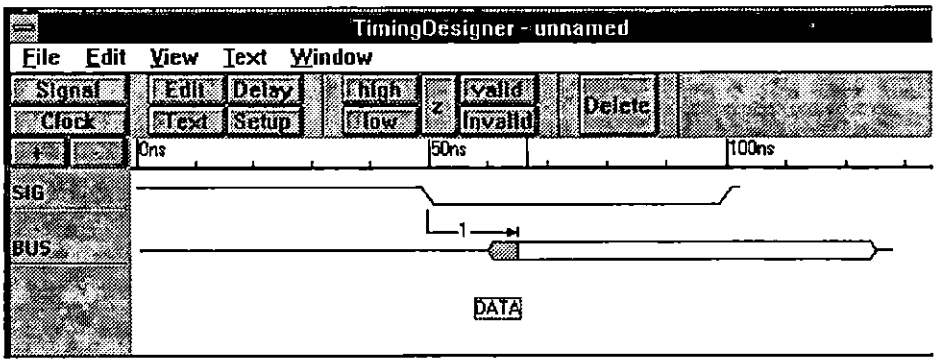


Every delay in a timing diagram (like the delay labeled 1, above) has a row in the Parameter Spreadsheet (1D).

Add an annotation

Annotations are used to add important comments to diagrams. You can add text anywhere in a timing diagram. In this exercise you will work with a text annotation, and use the SmartSnap feature.

1. Enable the SmartSnap feature by selecting the SmartSnap command from the Text menu.
2. Click the Annotation button. The Text tool appears (it looks like a crosshair). Position the tool below the BUS waveform, to the right of the delay, and then click. A rectangle appears. This is an empty annotation box.
3. An insertion point blinks inside the text box, ready for you to enter text. Type the word **DATA**.
4. Click the Pointer button for the Pointer tool. Because SmartSnap is on, the annotation is automatically attached to the nearest edge (notice the "string" that runs from the annotation to the edge on the left), and aligns itself with the waveform (in the Valid portion of the waveform BUS).



The example above shows the text field DATA before it is automatically positioned in the Valid segment of BUS. The "string" is not shown.

Examine attachments

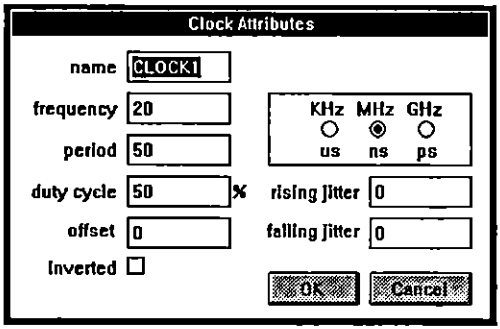
Annotation attachments are powerful because they allow you tie together an event and its descriptive text. In the previous exercise you saw how SmartSnap automatically attached the annotation to the nearest edge with a "string" and aligned the text to the signal. In this exercise, you'll examine attachments more closely.

1. Click the Pointer button for the Pointer tool. Drag the edge that is the source of the delay (the edge at about 50ns in SIG) by holding down the left mouse button, and then moving the mouse. The Edge Move tool (a horizontal double-headed arrow) appears, indicating the new location for the edge. When you release the mouse button, the edge you are moving snaps to the new location, and the target edge (in BUS) follows. TimingDesigner automatically maintains relationships specified by delays. Notice also that the annotation moves with the edge to which it is attached.
2. Now drag the annotation DATA. As you move it around the diagram, the "string" snaps to the nearest edge. This is a function of SmartSnap.
3. Release the left mouse button near any edge. You'll see that the annotation automatically aligns with the waveform. Experiment with different placements.
4. Experiment with the Detatch from Edge command by selecting the annotation DATA and then selecting the command. Since all annotations must be attached to some edge, you'll notice that the string jumps far to the left. The annotation is attaching to the invisible edge at time 0. Because this edge never moves, your annotation is now stationary.
5. Experiment with the Attach to Edge command by selecting the annotation and then selecting the command. The "string" attaches to the pointer. You can reattach the annotation by clicking on a new edge.
6. Before going on, drag the edge in SIG back to about 50ns, and relocate the annotation DATA to its former position.

Add a clock

Clocks are a special type of waveform. Clocks are defined using a dialog box, rather than by clicking with the mouse. This creates a waveform whose edges are regular and periodic. In this exercise, you will add a clock.

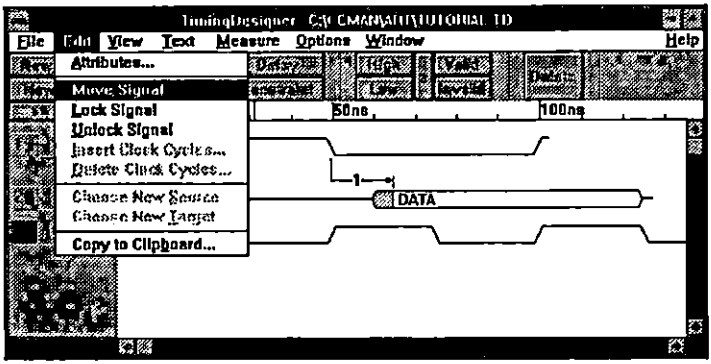
1. Click the New Clock button. The Clock Attributes dialog box appears with an insertion point in the Name field. Type **CLOCK1** in the Name field.
2. All of the other values in the dialog box are correct. Click OK. The Clock waveform appears at the bottom of the waveform area.



Move the clock

Although clocks are created and edited differently than signals, you can move both types of waveforms in the same way. In this exercise, you'll move the clock.

1. Select the Clock waveform by clicking its number, 3. The number turns dark gray to indicate it is selected.
2. Choose the Move Signal command from the Edit menu. A vertical double-headed arrow appears when the cursor is in the waveform name area.



The Move Signal command is used to move a selected signal to a new location.

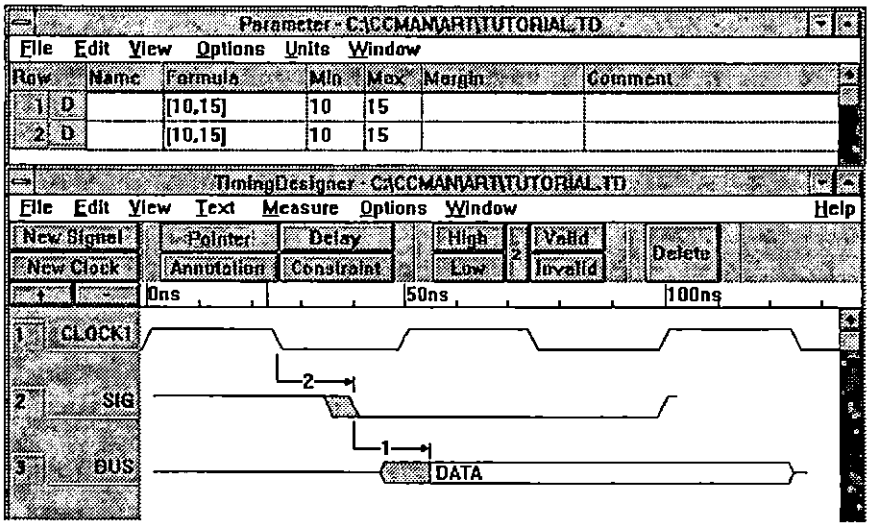
3. Position the arrow above the waveform name SIG, and then click. The Clock waveform moves to the top.

Add another delay

In this exercise you'll add another delay, and create a timing chain.

1. Click the Delay button. The Delay Source tool appears.
2. Click on the source edge, the first falling edge in CLOCK1 (at 25ns).
3. Click on the target edge, the first edge of SIG. A new row (2) is added to the Parameter Spreadsheet.
4. In the Parameter Spreadsheet, enter a Min of **10** and a Max of **15** for this delay.

Once you have entered the values, notice that not only does the uncertainty region appear on the target edge (on SIG), but the uncertainty on BUS expands. TimingDesigner automatically accumulates uncertainty as it propagates down a timing chain.



Delay paths are called timing chains. Timing chains whose source is a clock do not move.

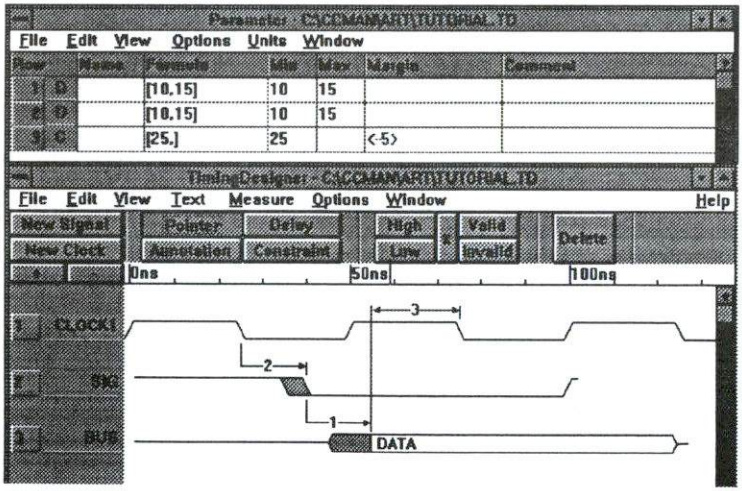
If you now try to move either of the edges in the chain, you won't be able to. This is because both edges are tied to the clock via delays. Since clock edges are locked by definition, the edges are stationary and won't move unless you change the clock parameters or delete the delays.

Add a constraint

Use the Constraint button to add timing constraints to the timing diagram. These can be setup and hold times or minimum pulse widths—any kind of measurement condition between two edges that must be maintained for the circuit to work.

1. Click the Constraint button. The Constraint Source appears. (This tool is similar to the Delay tool in that it has two halves: source and target portions.)
2. Set the source edge for the constraint. Click on the first edge at the start of data (at about 50ns in BUS).
3. Set the target edge. Click on the second falling clock edge (at 75ns). The constraint appears in green, indicating that the parameters of the constraint are satisfied.
4. Notice that a row, 3, is added to the Parameter Spreadsheet for this constraint. The C next to the row number indicates the row is a constraint. In the Name column, type **SETUP**.
5. In the Min column, type **25.** By entering this value, you are stating that the source edge must precede the target edge by a *minimum* of 25ns in order to work.

   In the Diagram Window, the green constraint turns red, indicating it is violated. The value **-5** in the Margin column indicates by how much the constraint is violated.



Timing constraints that must be maintained for the circuit to work.

Analyze and correct the constraint violation

One method of correcting the constraint violation is to use a slower clock. In this exercise, you'll redefine the clock.

1. To understand the violation, look at the measure. To determine actual measure, select the constraint (select the constraint in the diagram, or the row number in the spreadsheet), and then select the Show Measure command from the View menu. The label of the constraint now shows {20,30} in the Diagram Window.

   This indicates that the data on BUS is valid a minimum of 20ns before the target edge, 5ns less than required. Notice that the BUS data could be valid as early as 30ns before the target edge. This circuit therefore could work with fast parts, but fail with slow parts.

2. Click the Pointer button for the Pointer tool.

3. Double-click on the waveform number of the clock. The Clock Attributes dialog box appears.

4. Edit the Frequency field. Change it to from 20 (Mhz) to **16** (Mhz), and then click OK. The Clock waveform is modified. Whenever any change is made, TimingDesigner redraws the diagram to scale, validates all constraints, and recalculates the margins.

   The constraint is now green, meaning that it is satisfied. In the Parameter Spreadsheet, the margin changes to 7.50, and in the diagram, the measure changes to {32.50,42.50}.

Parameter - C:\CCMAN\ART\TUTORIAL.TD

File Edit View Options Units Window

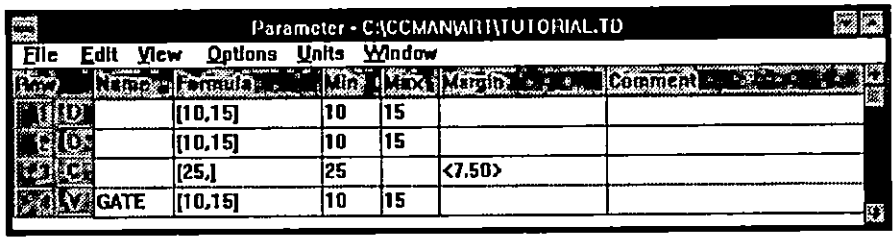
Row		Name	Formula	Min	Max	Margin	Comment
1	D		[10,15]	10	15		
2	D		[10,15]	10	15		
3	C		[25,]	25		<7.50>	

The Margin column shows how much a setup or constraint is violated or satisfied.

Add a variable

The Min/Max values we have used are identical (see the Min/Max values in rows 1D and 2D). This might occur when the same type of gate is used throughout a design. In cases like this, a variable can be used. In this procedure, we'll name the variable GATE.

1. In the Parameter Spreadsheet, select the Add variable command from the Edit menu. An empty variable (4V) appears at the bottom of the Parameter Spreadsheet.
2. Click in the Name column and type **GATE.**
3. Type **10** for the Min value and **15** for the Max.



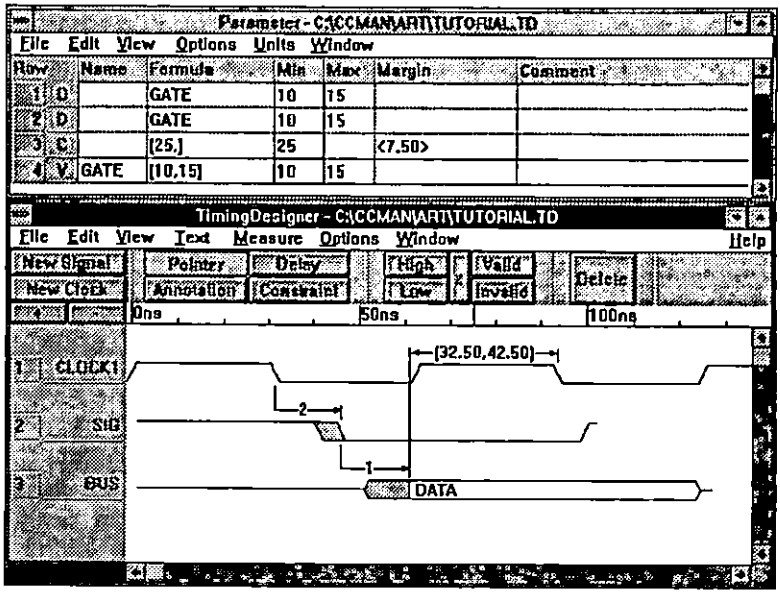
Row	Name	Formula	Min	Max	Margin	Comment
1D		[10,15]	10	15		
2D		[10,15]	10	15		
3C		[25,]	25		<7.50>	
4V	GATE	[10,15]	10	15		

4. Select the contents of the Formula field of delay 1 (Row 1) and replace it with **GATE.** Repeat for delay 2 (Row 2).
5. Using a variable allows you to easily perform some complex steps. For instance, suppose you wanted to experiment with propagation delays. Notice that the margin is currently 7.50. What would happen to that if an 18ns propagation delay (instead of the current 15ns) was used?

   Select the Max value of the variable GATE (15ns), and change it to **18.**

   The edges that use this variable are repositioned, and the margin is recalculated to 1.50. This means that a slower part (18ns) could be used without compromising the design.

The changes made in these steps to the timing diagram and Parameter Spreadsheet are illustrated on the next page.



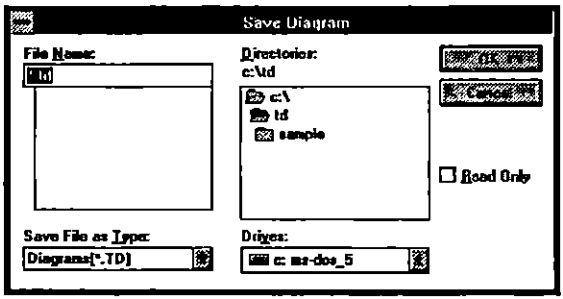
This is the final timing diagram and Parameter Spreadsheet.

Exit TimingDesigner

When you have finished with the tutorial, you can exit TimingDesigner.

1. Select the Exit command from the File menu. If you have unsaved work, you will be asked if you want it saved. Click Yes.

2. If you've named the timing diagram in progress, it is saved and you exit from TimingDesigner. If you have not named the timing diagram, the Save dialog box appears. Verify that the drive and directory choices are correct (change them using their respective fields), and then enter the file name in the File Name field.



3. Click OK. The file is saved under the name you entered, and you exit TimingDesigner.

Before Moving On...

If you've worked the tutorial, you've seen just a little of the power of TimingDesigner. In combination with the diagram you've created, use the rest of the chapters of this manual to experiment further. For instance:

- Print your timing diagram and spreadsheet, or export the data to other applications. See Chapter 12, "Printing and Exporting".
- Learn about TimingDesigner features that help you work efficiently. See Chapter 13, "Other Features and Functions".
- Begin inputting your own designs.

4

About TimingDesigner

This chapter introduces you to the concepts behind TimingDesigner and gives you an overview of the terms used. The topics included in this chapter are:

- the TimingDesigner windows
- timing diagrams
- waveforms
- delays
- constraints
- TimingDesigner files

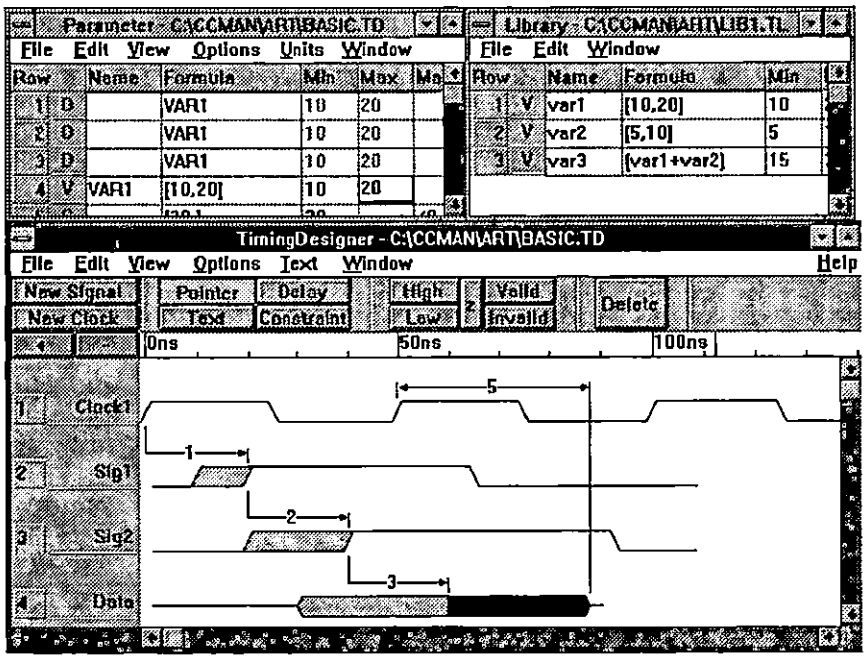
Some words in this chapter appear in *italics*. These are important TimingDesigner terms you should pay special attention to. The last section of the chapter contains a summary of them.

Working with TimingDesigner

Interaction with TimingDesigner takes place in three windows.

- In the Diagram Window you draw timing diagrams.
- In the Parameter Spreadsheet you specify the Min/Max values for the delays, constraints, variables, signal skews, and formulas that describe your timing diagram.
- In the Library Spreadsheet you specify common Min/Max variables, similar to an on-line databook, which may then be used in many designs.

In the illustration below, the three windows are Tiled. In the upper-left corner is the Parameter Spreadsheet, in the upper-right corner is the Library Spreadsheet, and running across the bottom is the Diagram Window.

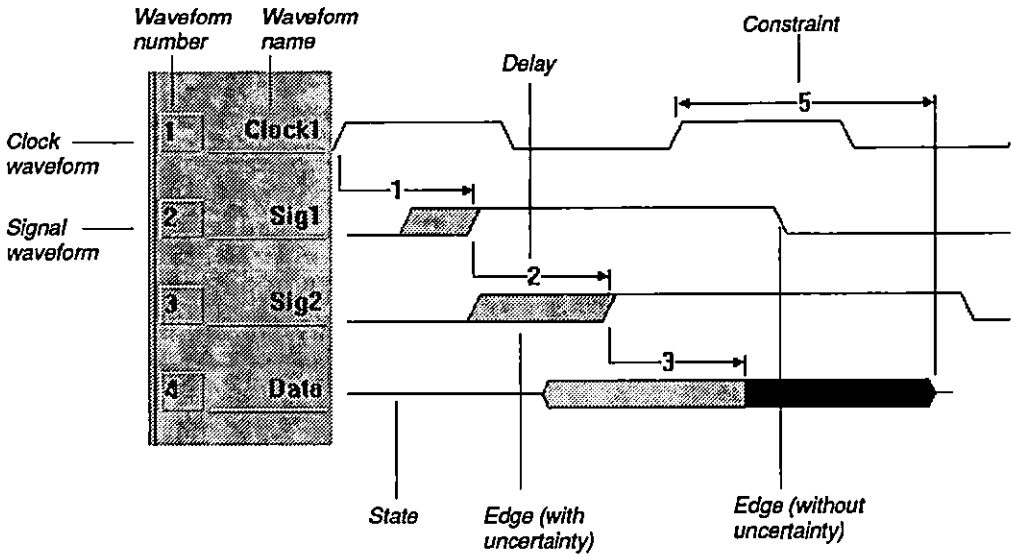


Comprehensive information on working with the windows, and their features and functions, is contained in Chapter 5, "TimingDesigner Basics."

Timing Diagrams

A timing diagram is a graphical representation of a design that shows the sequence of and timing relationships between the signals and edges of the design.

The illustration below shows a TimingDesigner timing diagram. The callouts indicate the various parts of the timing diagram, which are discussed in the following sections.



Timing diagrams use a number of graphic elements to portray timing relationships.

Waveforms

A waveform is the graphical representation of a signal in the circuit. Waveforms are composed of edges and states that describe the electrical functioning of an electrical signal in a circuit over time.

All waveforms have names. TimingDesigner names each waveform by default. You can change the default name anytime you wish, but names must be unique within a design.

TimingDesigner supports two types of waveforms: signal and clock.

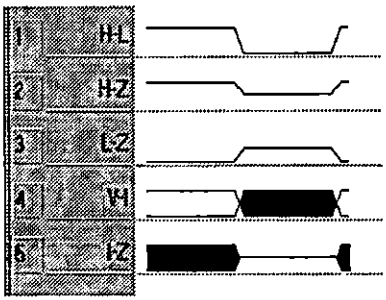
Signals Signals are drawn with the mouse in the Diagram Window, and are an ordered sequence of states and edges. Signals describe data, logic, or control signals within a circuit.

Clocks Clock waveforms are a special type of waveform. They are regular and periodic, and created via a dialog box, rather than by clicking with the mouse.

Working with waveforms is covered in Chapter 6, "Waveforms."

Edges and States

Waveforms are composed of two elements: edges and states. (Edges and states are covered completely in Chapter 7, "Edges and States.")



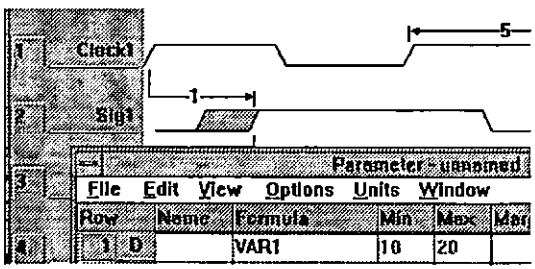
Some common state/edge transitions.

Edges

An edge is a transition from one state to another in the waveform. Edges are named for the state to which they transition. For instance, an edge between an Invalid state and a Valid state is called a Valid edge.

Uncertainty

In the real world, the exact time at which an edge occurs is rarely known exactly. Rather, an edge has an earliest and latest expected arrival. This time span is known as *uncertainty*. The uncertainty of an edge is represented by a gray area.



The uncertainty (gray region) of the first edge in Sig1 is caused by different minimum and maximum values of the delay specified in Row 1 of the Parameter Spreadsheet (inset).

Uncertainty appears on an edge when it is the target of a delay that has Min/Max values assigned to it. The uncertainty indicates the earliest possible arrival (specified by the delay minimum) and the latest possible arrival (specified by the delay maximum) of the edge. In actual operation, the edge could occur anytime between these two points, depending on the actual parts used to build the circuit.

Edge Display

TimingDesigner is able to display edges in two ways: Unsloped edges appear as straight vertical lines; sloped edges appear as sloped lines. The difference is for presentation only—the calculations are exactly the same. For further information, see Chapter 7, "Edges and States."

States

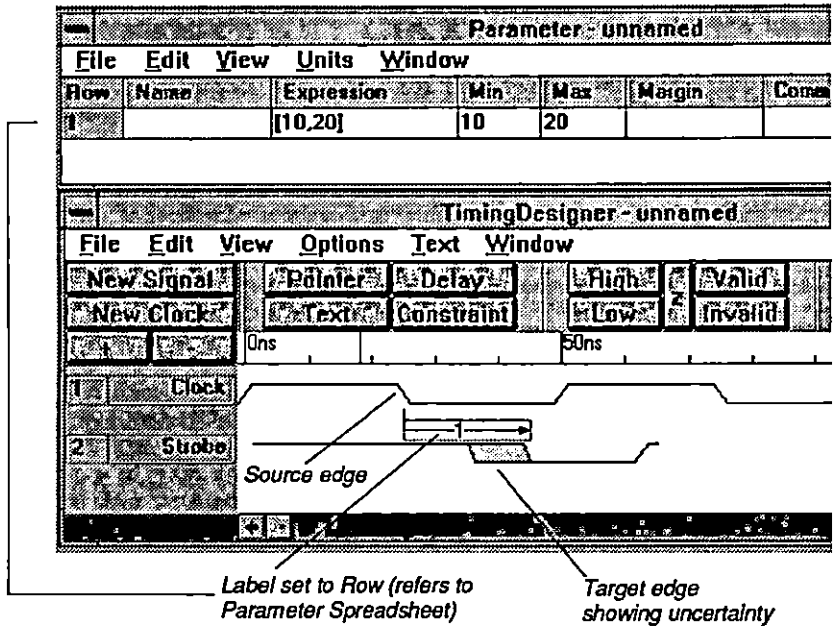
A state is the steady-state portion of a waveform between two edges. It represents the logic level of the signal at a point in time. TimingDesigner supports the following states:

High (H)
Low (L)
High-impedance (Z)
Valid (V)
Invalid (I)

Note that clock waveforms have only High and Low states.

Delays

A delay represents a cause-and-effect relationship between two edges, for instance, the propagation delay caused by a signal going through a logic gate. The *source* is an input signal transition, and the *target* is the resulting output signal transition.



In the example above, the earliest arrival of the target edge in Strobe is 10ns after the source edge (in Clock); the latest is 20ns after the source edge. These values are determined by entries in the Min and Max columns of the Parameter Spreadsheet (shown at the top of the illustration). These values are typically obtained from semiconductor data books, or from TimingDesigner libraries that you or others have created.

The most important concept to keep in mind when thinking about delays is that they "push edges around." The minimum and maximum values of a delay generate uncertainty for the target edge. When an edge that is a target of a delay is, in turn, used as the source for another delay, a *timing chain* is built. Along a timing chain, each edge has an uncertainty that indicates the accumulated uncertainty caused by all delays preceeding it in the timing chain. As you work with your design, these timing chains are maintained by TimingDesigner, so you never have to calculate the exact position of any edge in a design.

To summarize, every delay has two components:

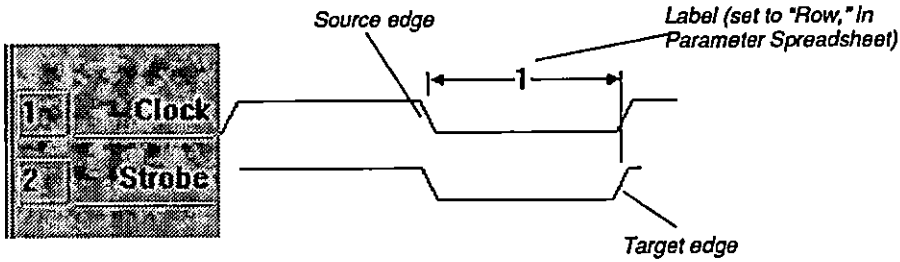
- The graphical representation in the Diagram Window, with lines and arrows indicating the source and target edges of the delay. A label is also displayed that identifies delay. The label can be changed.
- The numerical representation in the Parameter Spreadsheet, where the actual minimum and maximum values of the delay are recorded. The values can be actual minimums and maximums, or can be the result of using variables and formulas.

Working with delays is covered in Chapter 8, "Delays."

Constraints

A timing constraint establishes a relationship between two edges that must be maintained in order for the design to work. Setup and hold times are examples of constraints. Working with constraints is covered fully in Chapter 9, "Constraints."

Constraints are used to define and then test the distance between two edges. The two types of tests are *inside* and *outside*. An inside measurement is taken from the latest that the first edge can occur to the earliest that the second edge can occur. An outside measurement is taken from the earliest that the first edge can occur to the latest that the second edge can occur. Because these measurements are actually conditions which *must be met*, a constraint that fails the test is *violated*, and its representation in the timing diagram and its margin values in the Parameter Spreadsheet turn red (both are green, otherwise).



Constraints specify timing relationships that must be maintained for a circuit to work.

Like delays, constraints are also represented graphically in the timing diagram and numerically in the Parameter Spreadsheet (see the illustration below).

Margins

TimingDesigner automatically calculates the margin associated with each constraint. Margin indicates the amount of time by which the constraint is satisfied or violated. The margin value can be used, for instance, to guage whether a gate can be added or must be removed in a critical path. Margins are shown in the Parameter Spreadsheet.

Numerical Values

TimingDesigner offers comprehensive numerical abilities. You can create *variables*, and then combine those variables in *formulas* with mathematical *operators*. Working with numerical values is covered fully in Chapter 10 "Variables, Libraries, and Formulas".

All constants, variables, and formulas have both a minimum value and a maximum value. This is called a *Min/Max* that has the form [*minimum,maximum*] ([5,10] for example), because most values represent a time which is not known exactly.

A variable may be used in place of a Min/Max. For instance, if you have a gate delay of [5,10], you can name that delay GATE_DEL1, and use it wherever you might have used [5,10]. Should you choose a part with different specifications, you only have to change the value of GATE_DEL1 to recalculate the timing relationships in the entire design.

Formulas are combinations of variables and mathematical operators. For example, the formula below uses the variable mentioned above:

```
(2*GATE_DEL1)+[5,5]
```

Terminology

A list of terms introduced in this chapter is presented below. As you work with TimingDesigner, and as you use later chapters of this manual, refer here for information about the term.

Constraint	A measurement between two edges that defines a test that must be satisfied if the design is to work.
Delay	Typically, a propagation delay for a gate or a path delay for a trace.
Earliest	Describes the earliest arrival of an edge (the left-hand side of an uncertainty region). In formulas in this manual, earliest is notated as *e*. Mins are responsible for creating the earliest of an edge.
Formula	Numbers and variables combined with mathematical operators.
Inside	A constraint measurement from the latest of the source edge to the earliest of the target edge.
Latest	Describes the latest arrival of an edge (the right-hand side of an uncertainty region). In formulas in this manual, latest is notated as *l*. Maxs are responsible for creating the latest of an edge.
Min/Max	The basic form of evaluated numerical formulas (Min/Maxs, variables, and formulas) in TimingDesigner.
Outside	A constraint measurement from the earliest of the source edge to the latest of the target edge.
Source	The source edge of a delay or constraint. Source edges are the first edge "clicked-on" when adding a delay, and are often the input of a gate. A source edge is the causal edge. In formulas in this manual, source is notated as *s*.
Target	The target edge of a delay or constraint. Target edges are the second edge "clicked-on" when a adding a delay, and are often the output of a gate. A target edge is the "affected" edge. In formulas in this manual, target is notated as *t*.
Variable	A reference to a value that can be used in place of the value itself.

5

TimingDesigner Basics

This chapter tells you about working with TimingDesigner. The topics covered include:

- starting and exiting TimingDesigner.
- controlling TimingDesigner windows (using the Window menu).
- selecting objects.
- editing controls in the spreadsheets.

This chapter assumes an understanding of basic Windows concepts, such as selecting objects, editing text, and working with dialog boxes. Refer to your Windows documentation for complete information.

Starting and Exiting TimingDesigner

✔ To start TimingDesigner

1. Start Windows, if necessary.
2. Double-click on the TimingDesigner icon.

When TimingDesigner starts, the three windows appear cascaded (offset, one on top of the other), with the Diagram Window on top, *activated* (ready to be worked with). Information on opening a design or library is discussed in a later section, "Working With Files."

✔ To exit TimingDesigner

1. Activate the Diagram Window or Parameter Spreadsheet, if necessary.
2. Select the Exit command from the File menu.
   - If you have unsaved Library file data, you are asked if you want it saved. See the next section, "Working With Files."
   - If you have unsaved Diagram file data, you are asked if you want it saved. See the next section, "Working With Files."

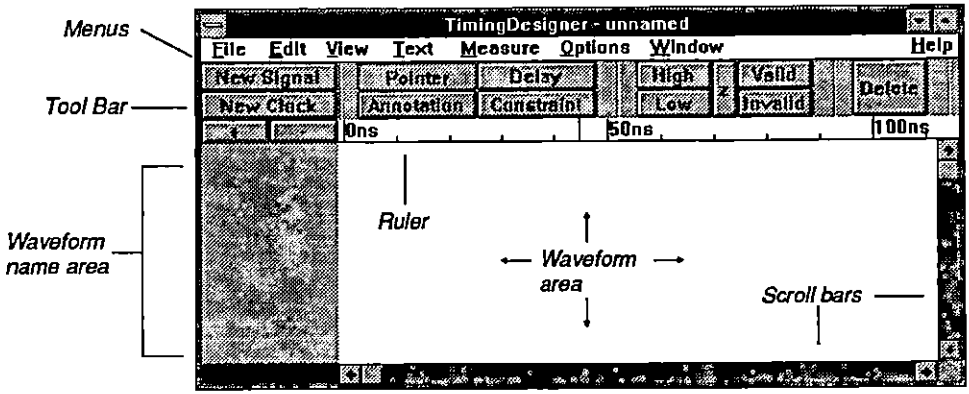
Once all prompts have been responded to, you exit TimingDesigner. All windows are closed and you are returned to the Windows desktop.

TimingDesigner Windows

The Diagram Window

The Diagram Window contains the graphical representation of circuit timing. Waveforms, delays, and constraints are drawn here.

Menus	The menus contain the commands that you use when creating timing diagrams.
Tool Bar	The tool bar provides instant access to the most frequently used commands and functions.



The Diagram Window is where you draw your timing diagrams.

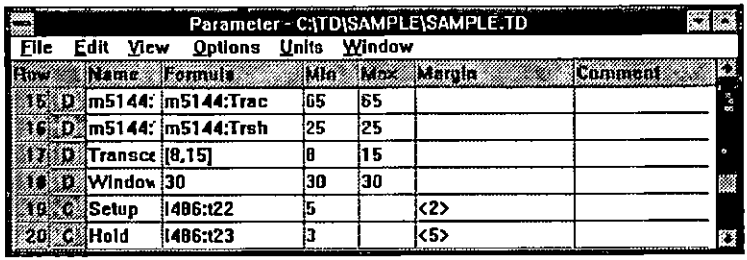
Waveform names	The gray field on the right of the Diagram Window holds the waveform numbers and names. Each waveform is labeled with a system-generated number that you cannot edit, and a waveform name that you can.
Ruler	The horizontal ruler assists in placing edges in your waveforms and making measurements.
Waveform area	The large white area is where you draw waveforms, delays, and constraints.
Scroll bars	The scroll bars allow you to move around your design.

Spreadsheets

To record numerical information about designs, TimingDesigner uses two spreadsheets: Parameter and Library.

The Parameter Spreadsheet

The Parameter Spreadsheet is similar to the parameter tables in a component data book, and contains information about the current design. Each row represents a timing parameter (a delay, constraint, or signal skew), or a variable. The columns within each row define the entry (Row number and type, Name, Formula, Min/Max, Margin, and Comment).


Parameter - C:\TD\SAMPLE\SAMPLE.TD
File Edit View Options Units Window
Row Name Formula Min Max Margin Comment
15 D m5144: m5144:Trac 65 65
16 D m5144: m5144:Trsh 25 25
17 D Transce [8,15] 8 15
18 D Window 30 30 30
19 C Setup I486:t22 5 <2>
20 C Hold I486:t23 3 <5>

The Parameter Spreadsheet. The Library Spreadsheet is similar to it, except that it has no Margin column.

Library Spreadsheet

The Library Spreadsheet is your on-line data book. It contains only variables, but these variables can be referenced by any number of designs. You can use the Library Spreadsheet to import Min/Max values into your Parameter Spreadsheet.

Working With Files

TimingDesigner produces two types of files: *Diagram files* and *Library files.*

Diagram files A Diagram file contains a timing diagram. It consists of the information in the Diagram Window and the Parameter Spreadsheet. Data in both of these windows is treated as a unit, and stored in a diagram file that has a default file name extension of .td.

Library files A Library file contains the variables in a Library Spreadsheet. Libraries have a default file name extension of .tl.

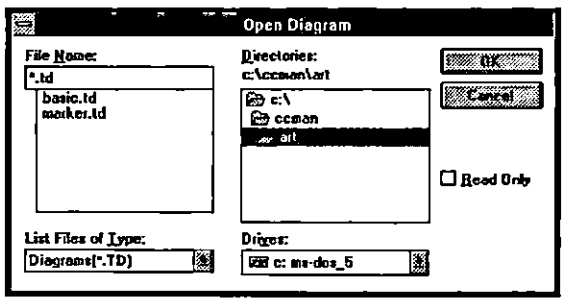
You work with Diagram files by using the commands on the File menu in the Diagram Window and Parameter Spreadsheet. You work with Library files by using the commands on the File menu in the Library Spreadsheet.

Opening a file

When you start TimingDesigner, all three windows appear cascaded. Use the Open command to bring a Diagram or Library file to the screen for editing. Diagram and Library files operate independently of each other, meaning that when you open a Diagram file, a Library file is not automatically opened.

✔ To open a file

1. Activate the window of the file type you want to open. For Diagram files, activate the Diagram Window or Parameter Spreadsheet; for Library files, activate the Library Spreadsheet.
2. Select the Open command from the File menu. The Open dialog box appears.



3. Locate the proper drive by using the Drives drop-down menu, and the directory by using the Directories list box.
4. Once the correct drive and directory have been chosen, choose one of the entries under List Files of Type field to filter the files to be displayed.
5. The File Name field reflects the filter you have chosen, and the list box shows all files in the directory that match. Open a file by double-clicking on the file name; or by selecting the file name, and then clicking OK. You can also type in the file name in the File Name field, and then click OK.

Clearing a file from the screen

Use the New command on the File menu to clear the current Diagram or Library file from the screen. If you have unsaved work, you are given the opportunity to save it. Saving work is discussed below under "Saving Your Work."

Saving your work

TimingDesigner allows you save your work when you exit the program, and at any time while working. Save your work often! In addition to simply saving your work, you can save files to new names.

✔ To save files

1. Activate the window that contains the data you want saved. For Diagram files, activate the Diagram Window; for Library files, activate the Library Spreadsheet.
2. Select the Save command from the File menu. If the Diagram or Library file already exists, it is saved immediately. If the work has never been saved, the Save As dialog box appears. (The Save As dialog box is similar to the Open dialog box.)
3. Verify that the drive, directory, and file information is correct, changing it where necessary.
4. Enter the file name in the File Name" field. You do not need to add a file name extension. The correct one is automatically appended for you.
5. Click OK.

✔ To save a file to a new name

1. Activate the window that contains the data you want saved. For Diagram files, activate the Diagram Window; for Library Files, activate the Library Spreadsheet.
2. Select the Save As... command from the File menu. The Save As dialog box appears (The Save As... dialog box is essentially similar to the Open dialog box; refer to the steps earlier for information on using the dialog box.)
3. Verify that the drive, directory, and file information is correct, changing it where necessary.
4. Enter the new file name in the File Name" field.
5. Click OK.

Controlling TimingDesigner Windows

You can control how TimingDesigner windows are displayed on your screen. The selections on the Window menu allow you to determine which windows are displayed and how they are positioned.

Selecting a window

The first three commands on the Window menu are the names of the TimingDesigner windows: Diagram Window, Parameter Spreadsheet, and Library Spreadsheet. Choose one of these commands to activate (or select) the named window. When you activate a window, it becomes the current window and moves to the top of the desktop. If the window was minimized (shrunk to an icon), it is restored.

✔ To activate a window, do any of the following

- Choose the Diagram Window command, or the Parameter Spreadsheet command, or the Library Spreadsheet command from the Window menu to activate the corresponding window.
- Click in any portion of a partially concealed window.
- Double-click on the icon of a minimized window.

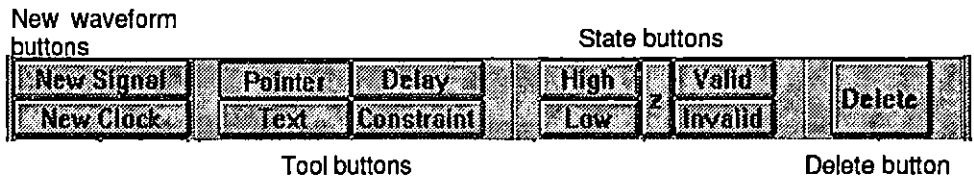
Arranging windows

Use the second group of three commands on the Window menu to arrange TimingDesigner windows:

Cascade	Sizes the windows to their default size, and then arranges them so that each title bar is visible. The order, from back to front, is Library Spreadsheet, Parameter Spreadsheet, and Diagram Window. If any of the windows were minimized, they are restored.
Tile (all)	Restores any minimized TimingDesigner windows, then sizes them to fill the entire screen. The Diagram Window is the largest, and fills the bottom two-thirds of the screen. The Parameter Spreadsheet and Library Spreadsheet share the top third of the screen. The Diagram Window is activated.
Tile (no Library)	Duplicates the Tile (all) command except that the Library Spreadsheet is excluded. The Diagram Window is activated.

The Tool Bar and Tools

In addition to pull-down menus, the Diagram Window features a tool bar. It contains command buttons that perform a number of important functions, including selecting the different tools.



The Tool bar buttons give you immediate access to important functions and commands.

The New waveform buttons

The New waveform buttons are used to add new waveforms to the design.

New Signal	Adds a new signal waveform.
New Clock	Adds a new clock waveform.

The Tool buttons

The Tool buttons select a tool to be used in the Diagram Window, and each button has at least one tool associated with it. For further information about tools, refer to the following section, "Tools: A closer look."

Pointer	Selects the Pointer tool. The Pointer tool is a general-purpose tool. It is used for placing and moving edges, selecting objects, and so forth.
Text	Selects the Crosshair tool that allows you to place and edit text blocks.
Delay	Selects the Delay Source tool used to add delays. When you click with the Delay Source tool on an edge, the pointer changes to the Delay Target tool.
Constraint	Selects the Constraint Source tool that allows you to add constraints. When you click with the Constraint Source tool on an edge, the pointer changes to the Constraint Target tool.

The State buttons

The State buttons are used to change the state of a selected state or edge. When an edge or state is selected, clicking one of these buttons changes the selected object to that state.

In addition, note that each of these buttons has a keyboard equivalent, such as "h" for High, "l" for Low.

The Delete button

The Delete button deletes a selected object. Unlike the Delete key, it is not used to delete individual text characters.

Tools: A closer look

You've already seen that some of the Tool Bar buttons have tools associated with them. But there are more tools than that available. The following table describes all the tools: what they look like, how they are selected, and what they do.

Tool Name	How accessed	Uses
Pointer	Click the Pointer button	The general-purpose tool: selects objects moves objects appends edges inserts pulses
Crosshair	Click the Annotation button	Determines the position of a new text block
Insertion Point (Diagram Window)	Appears after clicking in a selected text field with the Crosshair tool	Functions as the text cursor
Insertion Point (Spreadsheets)	Appears after clicking the Pointer tool in a field	Functions as the text cursor
Edge Move	Appears while dragging an edge that was selected with the Pointer tool	Moves edges

Tool Name	How accessed	Uses
Delay Source	Click the Delay button	Select the source edge of a delay
Delay Target	Appears after clicking with Delay Source	Select the target edge of a delay
Constraint Source	Click the Constraint button	Select the source edge of a constraint
Constraint Target	Appears after clicking with Constraint Target	Select the target edge of a constraint
Column Resize	Drag a spreadsheet column boundary	Resize the width of a spreadsheet column

Editing Text

The full range of Windows text editing features are available to you when working with text. Refer to your Windows documentation for comprehensive information about working with text and text fields.

Text in TimingDesigner occurs in two different forms, annotation text and entry fields.

Annotations Annotations appears in the a timing diagram, and are free-form, multiple-line fields used to annotate a diagram. Annotations are fully covered in Chapter 11, "Text Annotations."

Entry fields All other occurances of text are entry fields. Waveform names, cells in the spreadsheets, and entry fields in dialog boxes are examples of these. Note that some of these fields, such as the Name column in the spreadsheets, have special syntax requirements and restrictions on valid characters.

The Text menu in each TimingDesigner window contains Cut, Copy, Paste, and Clear commands for use with text.

Note: Cells in the spreadsheets have additional editing characteristics. See "Editing in the Spreadsheets," later in this chapter, for more information.

Working with Objects

Many interactions with TimingDesigner involve selecting an object, and then choosing an operation.

Selecting:How and what

The following table lists all of the objects that appear in a timing diagram, how to select the object, and how you know when an object is selected.

In general, TimingDesigner indicates selection either by changing an object to reverse video (in the case of text), or by surrounding it with a gray box called a selection box.

Object	Select by	Selection indicated when
Waveform	Clicking with the Pointer tool on the waveform number	The waveform number changes from light to dark gray
Waveform name	Clicking with the Pointer tool on the waveform name	The waveform name background changes to white, and an insertion point appears
Edge	Clicking with the Pointer tool on the edge	A light gray selection box surrounds the edge
State	Clicking with the Pointer tool on the state	A light gray selection box surrounds the state
Text blocks	Clicking with the Pointer tool or the Crosshair tool on the text block	A light gray selection box surrounds the text
Delays	Clicking with the Pointer tool on the delay label	A selection box surrounds the horizontal portion of the delay
Constraints	Clicking with the Pointer tool on the constraint label	A selection box surrounds the horizontal portion of the constraint
Spreadsheet row	Clicking with the Pointer tool on the row number	The row number turns dark gray
Spreadsheet cell	Clicking with the Pointer tool in the cell	The insertion point appears, and the cell expands if necessary

Selecting overlapping objects

Where objects such as delay or constraint labels and text fields overlap in the Diagram Window, the first click of the left mouse button selects one of the objects. Each subsequent click of the left mouse button selects an alternate object.

Selecting multiple graphic objects

You can select multiple graphic objects in the Diagram Window. You'll do this when, for example, you want to change the displayed label for a number of delays or constraints. Press Shift+Click to toggle the selection of multiple objects.

Selecting waveforms or spreadsheet rows

You can select multiple waveforms or spreadsheet rows.

Note: To select waveforms or rows, position the pointer over the system-generated waveform number (for waveforms) or on the row number (for spreadsheet rows).

Click	Selects a waveform or row.
Drag	Selects a range of waveforms or rows. Reverse direction to deselect.
Shift+Click	Creates a contiguous selection. Shift+Click on the first row of the range, and then Shift+Click on the last row of the range.
Ctrl+Click	Adds a single waveform or row to a selection.
Ctrl+Drag	Adds multiple waveforms or rows to a selection.

Working in the Spreadsheets

The Parameter and Library Spreadsheets are important elements of TimingDesigner. This section discusses what the spreadsheet columns are used for, and tells you how to control column widths.

Spreadsheet columns

If you look closely, you'll find that the only difference between the spreadsheets is that the Library Spreadsheet has no Margin column. The reason for this is that the Library Spreadsheet holds variables only, and the Margin column is used only by constraints. The following summarizes the spreadsheet columns.

Row/Type	The row number and type of entry. The number for a row is its ordinal position in the spreadsheet. Type indicates they type of information the row contains: D (delay), C (constraint), V (variable), S (signal skew).
Name	A one-word, free-form identifier. Names must conform to these rules:

All variable names must be unique. If you attempt to enter a duplicate name, TimingDesigner appends a numeral to the end of the name to differentiate it from the existing name.

All other names should be unique, but TimingDesigner will allow duplicates (for instance, a constraint and a delay might both be named FOO).

Valid names consist of any keyboard characters, including punctuation, but not space.

Formula — The algebraic forumula that is used to compute the minimum and maximum values.

If you enter a simple value, such as 15, in the Formula column, TimingDesigner automatically fills in the Min and Max columns with that value.

Min and Max — The values of a variable, delay, or constraint. Min/Max values are entered in one of two ways. If there is a formula in the Formula column, the system automatically calculates the Min/Max values, and they are not editable. If the Formula column contains a number (or is empty), these colums may be edited directly.

Margin — System-generated and non-editable, and is used only for constraints in the Parameter Spreadsheet. It shows the amount of time by which the constraint is either satisfied or violated. Positive values indicate that the constraint is satisfied; negative values indicate that it is violated. If either of the two values of a margin is negative, the constraint is violated.

Comment — A free-form field that may contain any information you choose.

Controlling spreadsheet column widths

Two special functions allow you to customize the way your spreadsheets look. First, you are able to resize all of the columns in the spreadsheets. Secondly, each spreadsheet Window menu contains a command to return the columns to their default size.

✔ To change column widths

- Position the Pointer tool over a vertical column separator near a column name, and drag. The Column Resize tool appears during the drag.

✔ To reset column widths to defaults

- Select the Reset Column Widths command from the Window menu in a spreadsheet. The column widths are returned to their default settings.

Editing in the spreadsheets

Each cell in a spreadsheet row is actually a text field. You can use standard Windows editing techniques with each cell. But spreadsheet fields differ from the text fields in dialog boxes. The following information applies specifically to spreadsheets.

- When editing a field that expands past its column width, the edit box expands. When you leave the field, the field returns to its preset size. The text you entered remains, although it cannot be seen. Select the field to re-expand it.
- Terminate a field edit and accept the changes you have made by pressing Enter, or click outside of the field.
- Abort a field edit by pressing Escape. The field returns to the state it was in before you entered it.
- Use the Cut, Copy, Paste, and Clear commands on the Edit menu (or use the Windows keyboard equivalents) to perform text edits of the fields.
- Press the Tab key to accept changes you make to a field and move the cursor to the next field.

Inserting a spreadsheet row

When you add a delay or constraint to a timing diagram, a row in the Parameter Spreadsheet is automatically created for the delay or constraint. When you want to create a variable or signal skew, however, you manually create the spreadsheet row.

✔ To create a spreadsheet row for a variable or signal skew

1. Activate the spreadsheet that is going to contain the variable or signal skew. (Signal skews are contained only in the Parameter Spreadsheet.)
2. From the Edit menu, select the Add Variable or Add Signal Skew command. A new row appears. You can now enter the values for the variable or signal skew.

Deleting a spreadsheet row

✔ To delete a spreadsheet row

1. Select the row by clicking on the row number.
2. Select the Delete command from the Edit menu.

   Note: Variables that are referenced in the Formula fields of other spreadsheet rows cannot be deleted.

   Note: Deleting the row of a delay or constraint in the Parameter Spreadsheet also deletes the corresponding graphic representation of the delay or constraint in the timing diagram.

6
Waveforms

Waveforms portray the working of your design. This chapter is about working with signal and clock waveforms.

Signals

Signals are waveforms that express the logic, data, or control signals in your design. The procedures below list the possible operations for signal waveforms.

Setting initial state

The initial state of a signal determines what the initial state is, and the toggle pattern for the signal (see below). If you don't set the initial state, the default initial state is High.

✔ To set the initial state of a waveform

When you add a signal waveform and the Signal Attributes dialog box appears, set the initial state.

Alternatively:

1. Select a signal.
2. Click one of the State buttons. The initial state will reflect the state you chose if the signal has edges.

About toggle patterns for new waveforms

When you add a new waveform, you specify the initial state. This automatically implies some kind of *toggle pattern* for edges as they are added.

For new waveforms, the pattern of these edges and states is predetermined, and based on the state of the first edge. The predetermined toggle patterns for new waveforms are:

H - L
L - H
V - Z
I - Z
Z - V

For example, if you want a to create a waveform that toggles between Z and Valid, set the initial state of the waveform to Z. The Z - V toggle pattern in the table above will be in effect.

For existing waveforms, the alternating pattern of the last two states is preserved with each new edge. For example, if the last two edges of a waveform are Valid and High, the first appended edge will be Valid, the second High, and so forth.

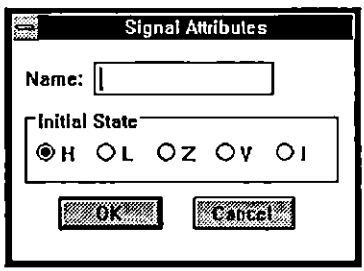
To change the toggle pattern, change the state of the edge just added. The new alternating pattern will be maintained.

Adding signals

New signal waveforms can be appended to the end of a timing diagram, or inserted at any point.

✔ To insert a new signal

1. Select an existing signal. The new signal will be inserted above this one.
2. Click the New Signal button. The Signal Attributes dialog box appears.



3. Enter a signal name in the Name field; and click one of the Initial State option buttons to specify the initial state. Click OK.
4. The new waveform appears above the waveform you previously selected. Its name is defaulted to the ordinal number of the waveform.

Note: You do not have to give a name to a waveform using the Signal Attributes dialog box. If you don't, the system will provide a name for you.

Note: You can suppress the appearance of the Signal Attributes dialog box by selecting the Skip Signal Dialog command on the Options menu. If you do this, TimingDesigner will assign a signal name. Use the State buttons to set or change the initial state.

✔ To append a new signal to the end of the diagram

Make sure that no waveform is selected, and then follow the steps above.

Deleting signals

✔ To delete a signal

1. Select the signal.
2. Click the Delete button.

Naming signals

Signals are named by the system when they are added. The default name is simply the ordinal number of the waveform. You can change the name to something meaningful to you. Signal names you create can consist of any number of alphanumeric characters. Each signal name must be unique.

✔ To change the name of a signal

1. Select the signal name.
2. Enter the new name, and then press Enter to confirm the edit; or press Escape to abort the edit.

Alternatively:

1. Double-click on the waveform number. The Signal Attributes dialog box appears (see description following).
2. Enter the new name in the Name field, and then click OK.

Moving signals

✔ To move signals

1. Select the waveform to be moved.
2. Select Move Signal from the Edit menu.
3. Indicate the new location for the waveform by selecting another waveform. The waveform to be moved is placed above the one you indicated.

Clocks

Clock differ from signals in two ways. First, clock waveforms are regular and periodic. Second, clocks are specified using a dialog box, rather than being drawn using the mouse. The procedures below list the possible operations for clocks.

Adding a clock

✔ To insert a clock in the middle of a diagram

1. Select a waveform. The clock will be added above this waveform.
2. Click the New Clock button. The Clock Editor dialog box appears (see description following).
3. Define the clock by entering the necessary parameters (see description following).
4. Click OK to accept the changes, or Cancel to abort the edit. The clock is inserted above the waveform you indicated.

✔ To append a clock at the end of a diagram

Make sure no waveforms are selected, and then follow the steps above.

Moving clocks

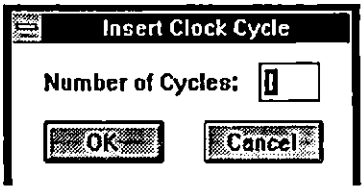
Clocks are moved in the same way as signals. Refer to the information on moving signals presented earlier in this chapter.

Inserting and deleting clock cycles

You can insert or delete clock cycles anywhere within a clock waveform. Inserting or deleting clock cycles has the effect of relocating all timing chain edges after the insertion/deletion point. Each clock cycle consists of two edges.

✔ To Insert clock cycles

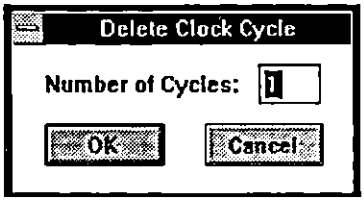
1. Select a clock edge.
2. Select the Insert Clock Cycles command from the Edit menu. The Insert Clock Cycles dialog box appears.



3. Enter the number of cycles to insert, and then click OK. The clock cycles are inserted earlier than the selected edge: this has the effect of "pushing" the selected edge and all later edges and their timing chains further out in time.

✔ To delete clock cycles

1. Select a clock edge.
2. Select the Delete Clock Cycles command from the Edit menu. The Delete Clock Cycles dialog box appears:



3. Enter the number of cycles to delete, and then click OK. The deletion begins at the selected edge, so all downstream timing chains appear to move back in time.

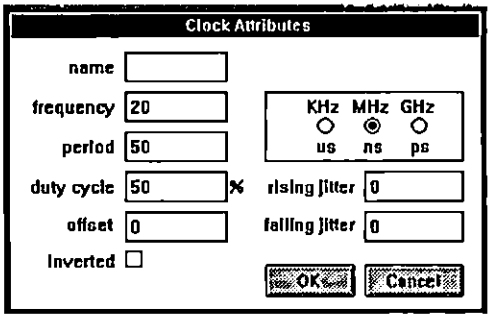
Note: If any of the clock cycles that you delete are the source of a delay, or the source or target of a constraint, the delay or constraint is also deleted.

Setting and Changing Clock Attributes

Clocks can only be edited through the Clock Attributes dialog box.

✔ To set or change the attributes of a clock waveform

Double-click on the waveform number of the clock waveform. Or, select the clock to be edited and then choose Attributes from the Edit menu. The Clock Attributes dialog box appears, and you can edit the waveform's attributes.



The Clock Attributes dialog box.

Clock Attributes dialog box fields

The following describes the fields in the dialog box.

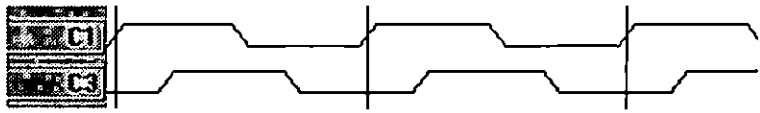
Frequency and Period — The length of a clock cycle. You are free to use either of these fields; an entry in one is automatically reflected in the other. Frequency is expressed in Hz (e.g., MHz). Period is expressed in fractions of a second (e.g., nanoseconds), according to the Unit button selected.

Duty cycle — The portion of the clock cycle in which the signal is high. Duty cycle is expressed as a percentage of the period.

These clock signals are identical except that C2 has a duty cycle of 25% as opposed to 50% on C1.

Offset

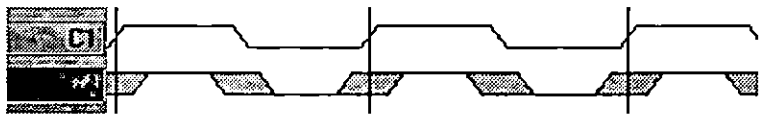
The location of the first transition within the clock cycle. Offset is used to shift the whole clock to the right. Offset is expressed in time units, according to the Unit button selected.



These clock signals are identical except that C3 has an offset value of 10ns.

Rising and falling jitter

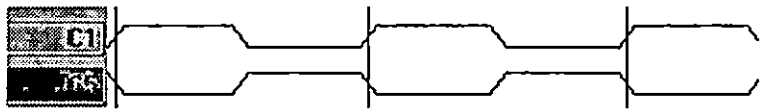
The uncertainty in the arrival time of the rising and falling clock edges, respectively. Jitter is specified in time units (e.g., nanoseconds), according to the Unit button selected.



These clock signals are identical except that C4 has a 10ns rising and falling jitter.

Inverted

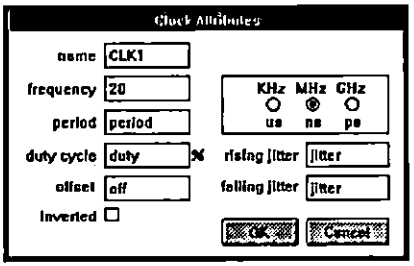
Checking this box changes the default state of a clock waveform from its default of Low to High.



These clock signals are identical except that C5 is inverted.

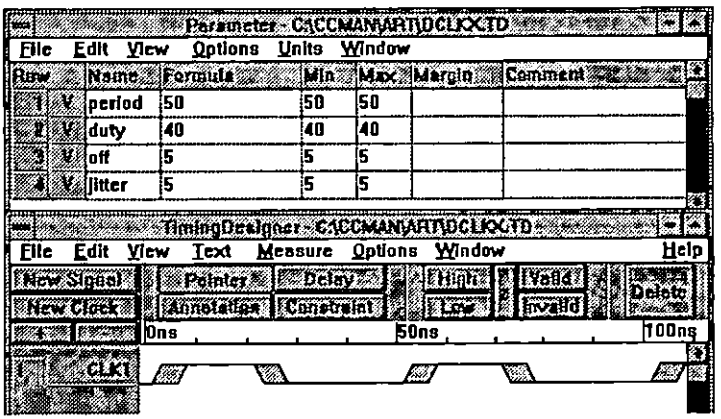
Using Formulas in Clocks

Clock parameters can be established by using formulas in the fields in the Clock Attributes dialog box. These formulas, consisting of variables and mathematical operators, allow you to define clocks that are easily changed (formulas are covered fully in Chapter 10, "Variables, Libraries, and Formulas"). The following example shows one way to use formulas in a clock.



As you can see, variables (simple formulas, in this case) are used in this clock for the Period (*period*), Duty cycle (*duty*), Offset (*off*), and Rising and Falling Jitter (*jitter*). This means that if the variables change, the clock changes automatically.

The resulting clock waveform and the Parameter Spreadsheet are shown below.

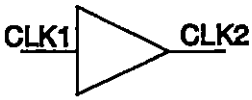


Creating Derived Clocks

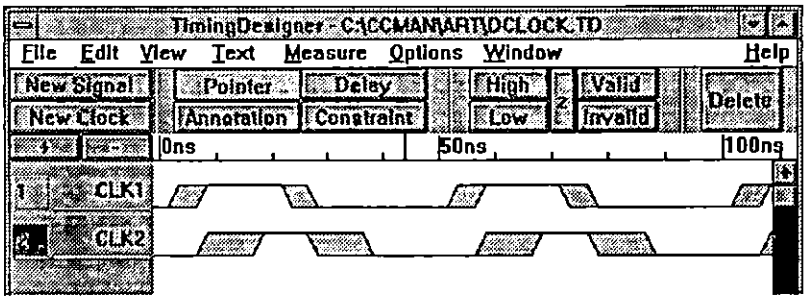
A derived clock is a clock waveform that is based in some way on another clock waveform. Using variables and formulas to specify the parameters of clocks (discussed in the previous section) makes it easy to derive one clock from another. It also ensures that if one clock changes, the other changes automatically. This section talks about deriving a clock by giving an example.

Derived clock example

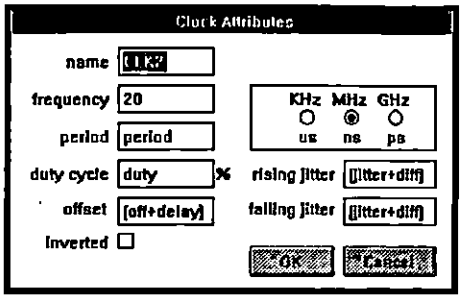
The type of clock being derived in this example is a clock waveform being put through a buffer that introduces a propagation delay of [5,10].



Shown below is a clock waveform (CLK1) and a clock derived from it (CLK2). (The Clock dialog box for clock CLK1 is shown in the previous section.)



The Clock dialog box for CLK2 shows how the clock parameters have been specified.



Two clock parameters are used to express the nature of the derived clock: Offset (which expresses the delay introduced by the buffer) and Rising and Falling Jitter (which express the uncertainty introduced by the buffer). The formulas for these two parameters are as follows:

- Offset is equal to the offset of the original clock (variable *off*), plus the average of the Min/Max for the delay:

  $$\frac{\text{Min} + \text{Max}}{2} + \text{off}$$

  which in this case yields $((\frac{5+10}{2}) + 5) = (\frac{15}{2} + 5) = (7.5 + 5) = 12.5$

- Rising and Falling Jitter is equal to the Max of the delay minus the Min of the delay:

  (Max - Min)

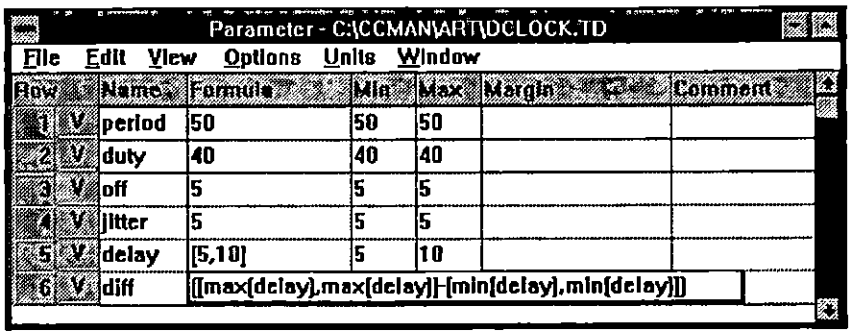
  which yields (10 - 5), or 5. This value is expressed by the variable *diff*. *Diff* uses Formula functions to compute its value. You can see the complete formula in the illustration below.

  The final calculation for Rising and Falling Jitter is to add the value of the variable *jitter* to *diff*:

  $5 + 5 = 10$

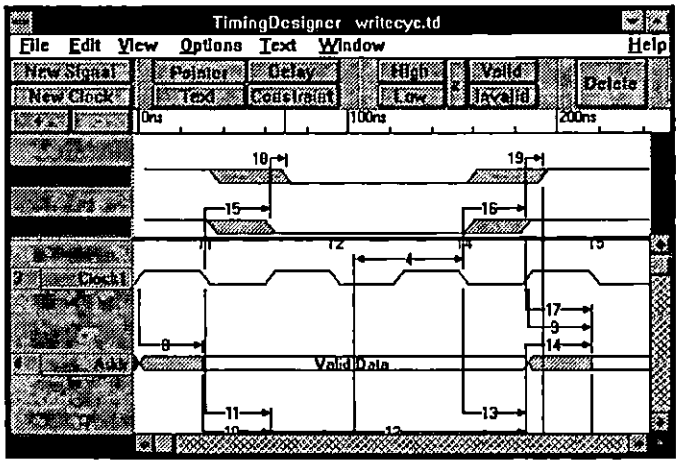
  Jitter is centered on an edge, so the uncertainty is 5ns on either side of each edge. This means that an the first rising edge of CLK2 has an uncertainty extending from 7.5ns to 17.5ns (12.5ns +/- 5ns). All subsequent edges have the same 10ns uncertainty.

The Parameter Spreadsheet showing all of the variables used is shown below.


Parameter - C:\CCMAN\ART\DCLOCK.TD
File Edit View Options Units Window
Row Name Formula Min Max Margin Comment
1 V period 50 50 50
2 V duty 40 40 40
3 V off 5 5 5
4 V jitter 5 5 5
5 V delay [5,10] 5 10
6 V diff [[max[delay],max[delay]]-[min[delay],min[delay]]]

Locking and Unlocking Waveforms

When you lock a waveform, you freeze it at the top of a timing diagram. Locked waveforms don't scroll vertically, so they are always visible. This allows you to select a number of waveforms and scroll the remainder design beneath them. The following illustration shows a timing diagram with locked waveforms.



An example of locked waveforms. The waveforms above the thick black line won't scroll vertically.

✔ To lock waveforms

1. Select the waveforms to be locked. The selection can be discontiguous.

2. Select the Lock Waveform command from the Edit menu. The waveforms are moved to the top of the diagram.

When you lock waveforms, a thick black line appears below the locked waveforms, and scroll bars are removed from the locked waveform area.

✔ To unlock waveforms

1. Select the waveforms to be unlocked. The selection can be discontiguous.
2. Select the Unlock Waveform command from the Edit menu. The waveforms move below the locked line. If you have unlocked all locked waveforms, the line disappears.

7

Edges and States

Edges and states are the two components that make up a waveform. This chapter discusses working with these components:

- how to add and edit edges, and to control edge attributes.
- how to add and control states.

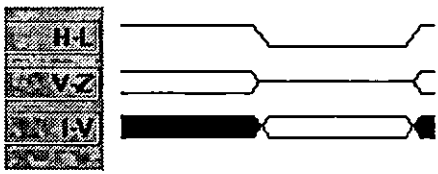
The information in this chapter applies only to signals, and not clocks. Clocks are controlled through the use of the Clock Attributes dialog box, and are not directly editable.

Working with Edges

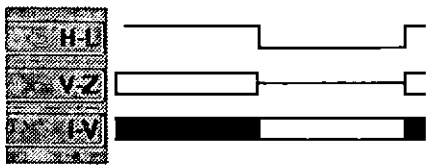
The most basic task in creating a timing diagram is indicating where the edges are. For this reason, understanding how to manipulate edges is important. This section discusses the operations possible on edges.

Using sloped or unsloped edges

TimingDesigner lets you choose how your edges are displayed, either as sloped or unsloped edges. Unsloped edges are drawn at right angles to states. The edge type you use makes no difference to TimingDesigner—it is a cosmetic decision. If your diagrams use short delays, or highly compressed time, unsloped edges may be better. The following illustrations show the edge types.



Common edge transitions using sloped edges.



Common edge transitions using unsloped edges.

Choosing the edge type is done on a diagram-wide basis using the Display Preferences dialog box, discussed in detail in Chapter 8, "Delays."

About toggle patterns

The toggle pattern of a waveform refers to the way the states alternate with each edge added. Toggle patterns are determined when a new waveform is added to the design. The predefined toggle patterns are:

H - L
L - H
V - Z
I - Z
Z - V

For example, if you want a to create a waveform that toggles between Z and Valid, set the initial state of the waveform to Z. The Z - V toggle pattern in the table above will be in effect.

For existing waveforms, the alternating pattern of the last two states is preserved with each new edge. For example, if the last two edges of a waveform are Valid and High, the first appended edge will be Valid, the second High, and so forth.

To change the toggle pattern, change the state of the edge just added. The new alternating pattern will be maintained.

See Chapter 6, "Waveforms," for more information.

Adding edges

Edges can be added to signals either at the end of the waveform or in the middle. When you add an edge in the middle of a waveform, you are actually adding a pulse—two edges and a state.

In most diagrams, edges need only to be placed approximately. The reason is that the final position for an edge most often depends upon timing relationships with other edges.

✔ To append an edge to the end of a waveform

1. Position the Pointer tool where you want the edge, using the hairline on the ruler face as a reference.
2. Click the left mouse button. An edge is added.

✔ To insert a pulse into a waveform

1. Position the Pointer tool at the desired location and double-click. A pulse of nominal width appears. The state of the pulse preserves the alternating pattern of the edges.
2. If necessary, you can change the state, and drag the edges.

Deleting edges

Any edge in a signal can be deleted.

✔ To delete an edge

1. Select the edge.
2. Click the Delete button.

   Note: When you delete an edge that is the source or target of a delay or constraint, the graphic is removed from the timing diagram, as well as the spreadsheet row that contains the name and Min/Max values for the delay or constraint.

Moving edges

Edges that have no restrictions placed on them (meaning edges that are neither locked nor are the source or target of a delay) are free to move left and right.

When an edge is part of a timing chain (a sequence of edges connected by delays, with Min/Max times specified), all edges in the timing chain move to preserve the Min/Max values of all the delays involved. The only time this is not true is if a clock is the source of the timing chain.

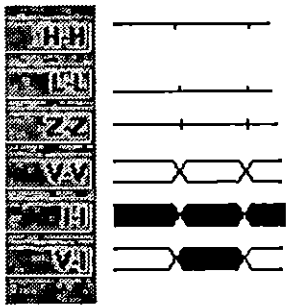
Edges that are not part of a timing chain will not move for two reasons: they are either locked or are edges in a clock. Locked edges are explained in a later section in this chapter.

✔ To move an edge

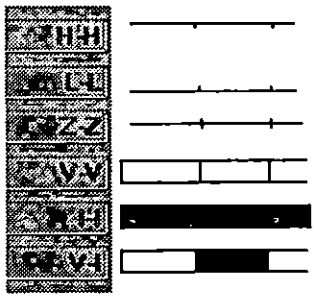
1. Using the Pointer tool, drag the edge. The Edge Move tool appears.
2. Release the left mouse button when the edge is in the desired location.

Changing edge state

When you change the state of an edge, you change the right-hand portion of the edge and the state following it to the new edge/state type. This can sometimes be visually confusing when the change results in a transition from one state to the same state. The illustrations below shows state-to-same-state transitions. The signal names indicate the states.



State-to-same-state edge transitions using sloped edges.



State-to-same-state edge transitions using unsloped edges.

The small tick marks (on the H-H signal, for example) indicate an edge exists in that location, but it is in the middle of a state-to-same-state transition.

✔ To change the state of an edge

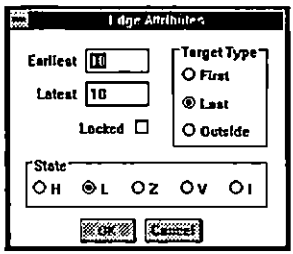
1. Select the edge.
2. Click the appropriate State button, or press the right mouse button to cycle through possible edge states.

Setting and Changing Edge Attributes

Use the Edge Attributes dialog box to control the functions and features of edges.

✔ To set or change edge attributes

Double-click on an edge. Or, select the edge and then select the Attributes command from the Edit menu. The Edge Attributes dialog box appears.



Edge Attributes dialog box fields

Earliest and Latest — Specifies the earliest and latest occurance of the edge. When these values are different, an uncertainty results. (The width of the uncertainty region depends upon the delays driving this edge.) If one field is changed, the other automatically updates to maintain the width of the uncertainty region. If there is no uncertainty region, then the values are always identical. Locking the edge (see section following) allows you to fix the edge at specific times.

Locked — Locks the edge to the current point in time (neither delays nor the mouse can move the edge). More information on locking edges is contained in the following section.

Target Type — Affects only those edges that are the target of more than one delay. A complete descripition of how to choose target types is contained in Chapter 8, "Delays."

State — Sets or changes the state of the edge.

Locking Edges

Once an edge is locked, it no longer responds to changes in the timing of your design. Locked edges cannot be moved in any fashion until they are unlocked. Locking also allows you to:

- Place an edge arbitrarily.
- Create an arbitrary uncertainty on an edge.
- Create a simple, graphic picture of timing, rather than a fully functional timing diagram.

Locking edges is an escape hatch that defeats the automated edge-placement features of TimingDesigner. Use it cautiously!

✔ To lock an edge

1. Double-click on the edge to be locked. The Edge Attributes dialog box appears (see the description of the Edge Attributes dialog box, earlier in this chapter).
2. Check the Locked check-box.
3. Enter values in the Earliest and Latest fields. If the values are different from each other, the edge will show uncertainty.

Using Snap to Place Edges

The snap feature enables you to place signal edges at precisely the same time as the edges of a specified clock. Snap works by creating an invisible grid based on the edges of the selected clock. When Snap is on and you add an edge, if the Pointer tool is positioned approximately below a clock edge, the edge you add will snap to the same time as the edge in the selected clock waveform. The window for Snap is equal to the width of a sloped edge.

✔ To enable Snap

1. Select a clock waveform.
2. Select the Snap command from the Options menu.
3. Create all the edges that you want snapped to the clock waveform.

States

States are the steady-state part of the signal between edges. Working with states is fairly simple, because there are few operations to perform on them.

Deleting states

Any state in a timing diagram can be deleted. What happens when you delete a state is that the state and the edge on the left of the state is removed from the drawing.

✔ To delete a state

1. Select the state.
2. Click the Delete button.

Note: Deleting a state bounded by an existing edge may result in an unusual picture.

Changing states

✔ To change a state

1. Select the state.
2. Click one of the State buttons; or, cycle through states by clicking the right mouse button.

Note: Changing states may result in an unusual picture.

8
Delays

Delays form the foundation of timing in timing diagrams. The topics discussed in this chapter include:

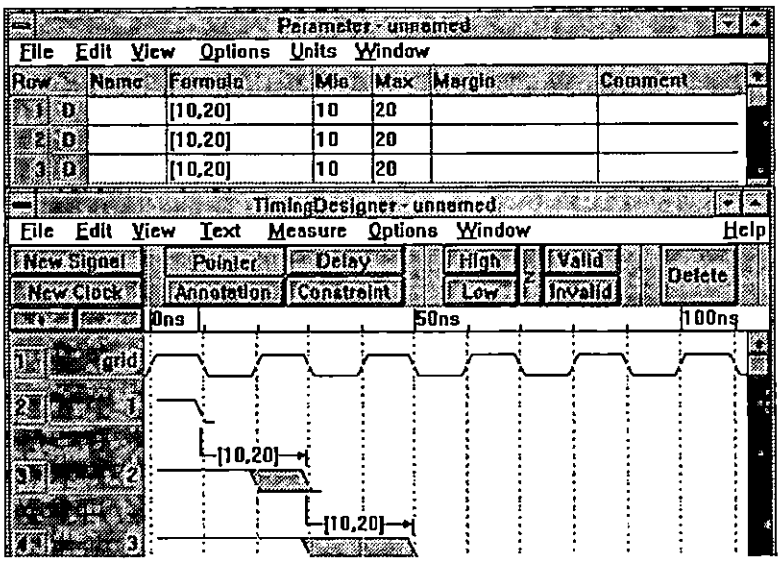
- a description of how delays function.
- how to work with delays—adding, editing, entering numeric values.
- setting labels and preferences, and creating defaults.
- how to work with multiple delay types.

To get the most use out of the information in this chapter, you'll need to be familiar with a number of terms introduced in Chapter 4, "About TimingDesigner." Turn there as necessary, paying special attention to the "Terminology" section.

About Delays

A delay represents a propagation delay and implies a causal timing relationship between two edges. The relationship consists of two parts: the delay in the timing diagram that indicates the source and target edges and the Min/Max parameters entered in the Parameter Spreadsheet. Looked at another way, in the timing diagram you indicate the edges involved, and in the Parameter Spreadsheet you specify the difference in time the target edge will trail the source edge.

Once both parts of a delay are in place, the relative position of the two edges are automatically maintained. This means that when you move an edge that is the source or target of a delay, edges associated with that edge also move. This is how timing chains are built. As you work with your design, these timing chains are maintained by TimingDesigner, so you never have to calculate the exact position of any edge in a design. The following illustration shows how delays build a timing chain.



The illustration above also shows some other interesting features:

- Using the grid lines for a reference, you can see how the delay times accumulate. The Min/Max of each delay is [10,20].
- Minimums always combine with minimums to affect the earliest possible arrival of the target edge, and maximums with maximums to affect the latest possible arrival of the target edge.

- The values for each delay are recorded in the Parameter Spreadsheet (shown at the top of the illustration).

Delay formulas

How delays work can be easily defined.

t.e = s.e + Min
(target earliest = source earliest + Min of the delay)

t.l = s.l + Max
(target latest = source latest + Max of the delay)

Referring to the previous illustration, the calculation for the earliest arrival of the target edge is equal to the earliest arrival of the target edge plus the minimum value of the delay. Using Signal 2 as an example:

t.e = (20ns + 10ns) = 30ns

Referring again to Signal 2, above, the calculation for the latest arrival of the target edge is equal to the latest arrival of the target edge plus the maximum value of the delay.

t.l = (30ns + 20ns) = 50ns

Working with Delays

Adding a delay

✔ To add a delay to the design

1. Click the Delay button. The Delay Source tool appears.
2. Click on the source edge of the delay. The source edge is selected, and the Delay Target tool appears.
3. Click on the target edge of the delay. The target edge is selected, and the system draws the delay. The Delay Source tool returns.

Immediately after adding a delay, you can do one of the following:

- Add another delay.
- Change to another tool by clicking a tool button. (This also aborts the process of adding a delay.)

- Enter the Min/Max for the delay in the Parameter Spreadsheet.

Deleting a delay

✔ **To delete a delay, do one of the following**

- In the diagram, select the delay, and then click the Delete button.
- In the diagram, delete one of the edges of the delay.
- In the Parameter Spreadsheet, select the delay, and then select the Delete command from the Edit menu.

Entering numeric values for a delay

When you add a delay to a timing diagram the delay has no effect on its edges until numeric values are associated with it.

✔ **To add numeric values for a delay**

1. Activate the Parameter Spreadsheet, and locate the row for the delay.
2. Enter a formula, or fill in the Min/Max columns.

Alternatively:

1. Immediately after adding the target edge of the delay, click the Pointer button for the Pointer tool.
2. Double-click on the delay label. The Delay Attributes dialog box appears.
3. In the Formula field, enter a formula or a Min/Max (such as [5,10]), and then click OK. The values are automatically copied to the Parameter Spreadsheet.

   Note: The Delay Attributes dialog box is described in the next section.

Hiding a delay

✔ **To hide a delay**

1. In the diagram, select the delay by selecting its label.
2. Select the Hide command from the View menu. The delay disappears from the timing diagram.

Alternatively:

1. In the Parameter Spreadsheet, select the delay by selecting its row number.
2. Select the Hide command from the View menu. The row number in the Parameter Spreadsheet turns light gray to indicate the delay is hidden.

Alternatively:

1. Double-click on the delay label. The Delay Editor dialog box appears.
2. Check Hide, and then click OK. The delay disappears from the timing diagram, and the row number in the Parameter Spreadsheet turns light gray to indicate the delay is hidden.

Showing a hidden delay

✔ To show a hidden delay

1. Activate the Parameter Spreadsheet.
2. Select the row of the hidden delay by clicking on the row number.
3. Select the Show command from the View menu in the Parameter Spreadsheet. When you activate the Diagram Window the delay will have reappeared.

Alternatively:

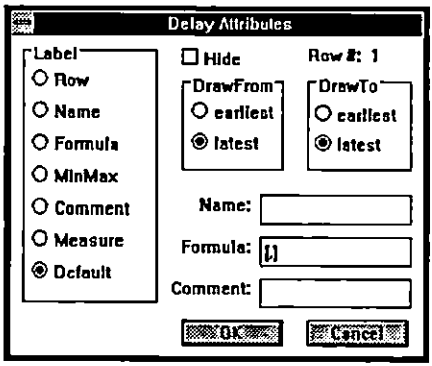
1. Select the Display Preferences command on the View menu. The Display Preference dialog box appears.
2. Click the Gray option in the Hidden field, and then click OK. All hidden items are now gray instead of invisible, allowing you to identify objects.

Setting and Changing Delays

You can edit all of the attributes of a delay by using the Delay Attributes dialog box.

✔ To edit a delay

1. Double-click on the delay label; or select the delay, and then select the Attributes command from the Edit menu. The Delay Attributes dialog box appears. The fields of the dialog box are explained below.



2. Set the attributes of the delay, and then click OK.

Delay Attributes dialog box fields

Label	These buttons are used to specify the label for the delay (for more information on labels, see the section, "Display Preferences dialog box fields" later in this chapter).
Hide	Checking this box hides the delay.
Row number	Lists the row number of this delay in the Parameter Spreadsheet. This makes it easy for you to check on the row number for this delay when the label is set to something other than Row.
Draw From and Draw To	These fields allow you to set how the delay is graphically displayed. The Draw From options determine where the source is indicated, and the Draw To options determines the target.
Spreadsheet fields	The Name, Formula, and Comment fields contain entries for this delay from the Parameter Spreadsheet. If you edit these fields here, the edits are copied to the Parameter Spreadsheet.

Setting and Changing Display Preferences

The display preferences that you set in TimingDesigner control the following:

- The label. The label is the text in the Diagram Window that identifies a delay or constraint. Every delay or constraint in a timing diagram has a label, and you can choose its type.
- Whether edges are sloped.
- Whether uncertainties are shown.

Note: Because of the similarity of the operation, this section discusses labels and preferences for both delays and constraints.

Setting and changing global defaults

Using the Display Preferences dialog box, you can establish global defaults for label type for all new delays and constraints. In addition, the dialog box also allows you to set two other global viewing parameters, sloped edges and uncertainties.

✔ To set global defaults

1. Select the Display Preferences command from the View menu in the Diagram Window. The Display Preferences dialog box appears (described later in this section).
2. Indicate the default state for Labels, Edge display, and Uncertainties.
3. Click OK to accept the new defaults. All waveforms, delays, and constraints added after these choices are made will reflect these choices.

✔ To make an exception to global labeling

Once you have determined the global setting for labeling delays and constraints, you are free, on a case by case basis, to change a delay or constraint's label. This allows you to tune how your timing diagram appears.

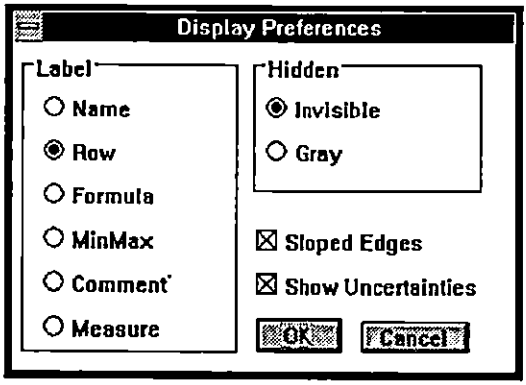
1. Select the delay or constraint label to be changed.
2. On the View menu, choose one of the "Show *label*" commands.

✔ To remove an exception to global labeling

If you want the label of a delay or constraint always to reflect the global setting, select the delay or constraint in the Diagram Window or the Parameter Spreadsheet, and then choose the Show Default command from the View menu in the respective window.

Display Preferences dialog box fields

This section summarizes the fields in the Display Preferences dialog box (shown below). The label definitions also extend to the Show commands on the View menu.



Labels

When you click one of the radio buttons in the Label field, you set the global condition for the labels of all new delays and constraints added, and for all existing delays and constraints that are set to Default.

Name	The entry in the Name column of the Parameter Spreadsheet (e.g., GATE1DEL). If the delay or constraint does not have a name, then Row is used instead.
Row	The Row number in the Parameter Spreadsheet (e.g., 4).
Formula	The contents of the Formula cell in the Parameter Spreadsheet.
Min/Max	The Min/Max value of the delay or contraint in the Parameter Spreadsheet (e.g., [10,20]).
Margin	The value in the Margin column of the contraint in the Parameter Spreadsheet (e.g., <10>, or <-15> for violated margins). This choice appears only for constraints.

Measure The inside and outside distance between the source and target edge (e.g., {12,15}).

Comment The text in the Comment field for this delay or constraint in the Parameter Spreadsheet (e.g., DATA HOLD). If the Comment field is empty, the delay or constraint is not labeled.

In all cases, if the label chosen is too long to fit between the vertical legs of the delay or constraint, the label is moved to the left of the legs.

Hidden options

Invisible When the Invisible option is enabled, hidden delays and constraints are not visible in the timing diagram. The Row number of hidden delays and constraints in the Parameter Spreadsheet is grayed.

Gray When the Gray option is enabled, hidden delays and constraints appear grayed in the timing diagram. The Row number of hidden delays and constraints in the Parameter Spreadsheet is grayed.

Sloped edges option

When you check Sloped Edges, all edges are shown as sloped lines, rather than vertical lines.

Show uncertainties option

When you check Show Uncertainties, uncertainties are shown for all waveforms, if they are present.

Multiple Delays

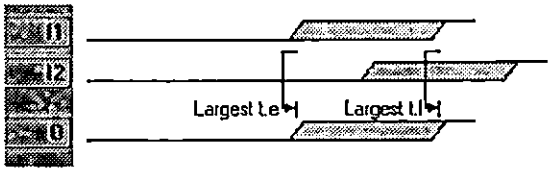
The introduction to delays at the beginning of this chapter covered the basic function of delays. Now we're going to look at a more complex type of delay structure: multiple delays with the same target (referred to simply as multiple delays).

Multiple delays to the same target occur when you are modeling a gate with multiple inputs (sources) and one output (target). How the target edge is positioned in these cases depends upon an attribute of the target edge called target type.

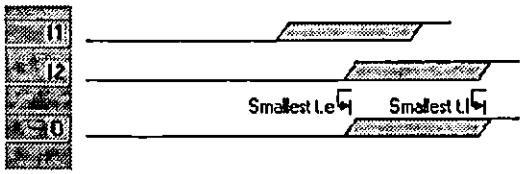
TimingDesigner offers three target types. The type you choose depends upon the function being described.

Target Types

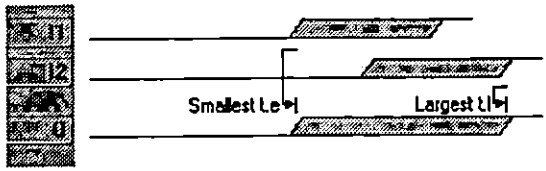
The three target types TimingDesigner uses are called first, last, and outside. These edge are roughly equivalent to the operation of OR, AND, and XOR, gates, respectively. However, it is a rough equivalency. The following illustrations depict in timing diagram format multiple delay types. Use these illustrations to match a target edge type to the timing you are describing.



An example of the critical paths for the target type first.
This is like the assertion of an OR gate.



An example of the critical paths for the target type last. This is like the assertion of an AND gate.



An example of the critical paths for the target type outside. This is *somewhat* like the assertion of an XOR gate.

Critical paths and the combination rule

The way TimingDesigner positions target edges when multiple delays are involved is to analyze the source delays, and then identify the critical paths. In the beginning of this chapter we presented the formula for how delays are calculated. To review:

t.e = s.e + Min
t.l = s.l + Max

These formulas are applied to each of the delays. Once the values are obtained, the the two critical paths are identified according to the *combination rule*. (The word *combination* in this sense refers to which mins and maxs are combined to position the target edge.)

The Combination Rule:

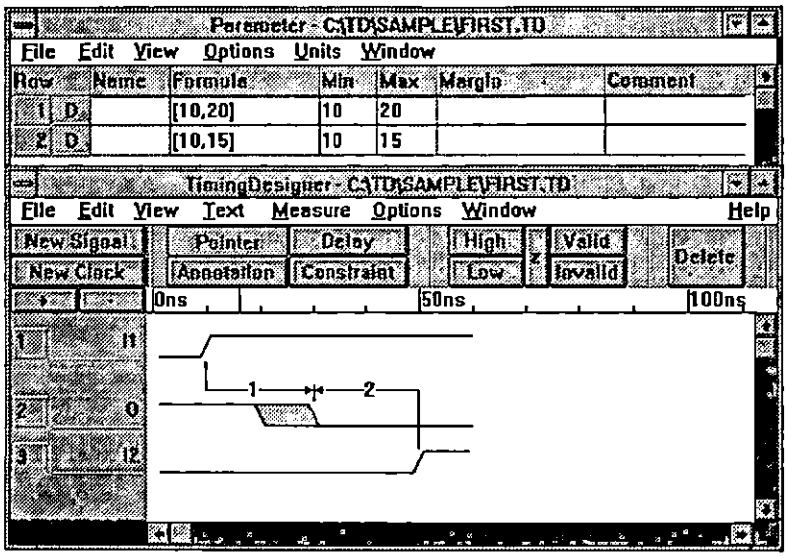
The critical paths used to position a target edge depend upon the target edge type.

	Combination Rule:	
Target edge type:	*Value of t.e*	*Value of t.l*
First	Smallest	Smallest
Last	Largest	Largest
Outside	Smallest	Largest

Note: In all cases, it is possible for one delay to be the critical path for minimum calculations, and another to be the critical path for the maximum calculation.

About target edge type first

The First edge type is used to describe logic whose timing acts like the onset of an OR gate. The illustrations below show a typical OR gate in schematic notation, and then the same gate as described by TimingDesigner.



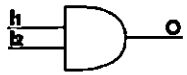
The timing diagram above can also be used to analyze a First edge type. To determine the critical paths, the delays are analyzed to find the earliest min and max at the target. For example:

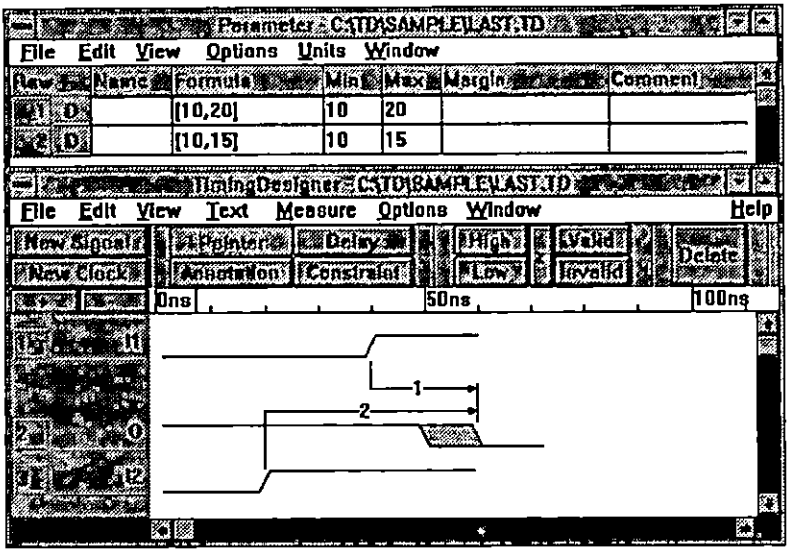
Minimum critical path:
1 = 10ns + 10ns = 20ns (critical path, minimum)
2 = 50ns + 10ns = 60ns

Maximum critical path:
1 = 10ns + 20ns = 30ns (critical path, maximum)
2 = 50ns + 15ns = 65ns

About target edge type last

The Last edge type is used to describe logic whose timing acts like the onset of an AND gate. The illustrations below show a typical AND gate in schematic notation, and then the same gate as described by TimingDesigner.





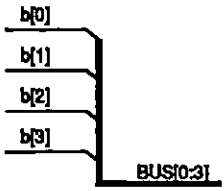
The timing diagram above can also be used to analyze a Last edge type. To determine the critical paths, the delays are analyzed to find the latest min and max at the target. For example:

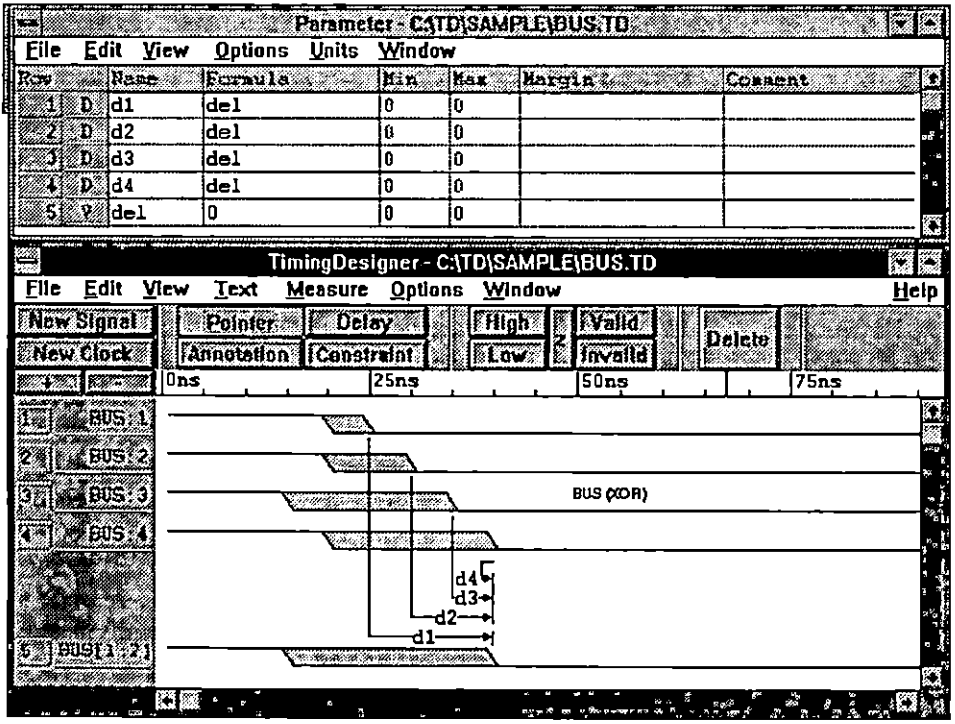
Minimum critical path:
1 = 30ns + 10ns = 40ns (critical path, minimum)
2 = 20ns + 10ns = 30ns

Miximum critical path:
1 = 30ns + 20ns = 50ns (critical path, maximum)
2 = 20ns + 15ns = 35ns

About target edge type outside

The third target edge type, Outside, can be used to represent the relationship between a bus and its constitutents. The following illustration shows a 4-bit bus in schematic notation, and then the same bus modeled in TimingDesigner.





The timing diagram above can also be used to analyze an Outside edge type. To determine the critical paths, the delays are analyzed to find the earliest min and and the latest max at the target. For example:

Minimum critical path:
d1 = 20ns + 0ns = 20ns
d2 = 20ns + 0ns = 20ns
d3 = 15ns + 0ns = 15ns (critical path, minimum)
d4 = 20ns + 0ns = 20ns

Maximum critical path:
d1 = 25ns + 0ns = 25ns
d2 = 30ns + 0ns = 30ns
d3 = 35ns + 0ns = 35ns (critical path, maximum)
d4 = 40ns + 0ns = 40ns (critical path, maximum)

9

Constraints

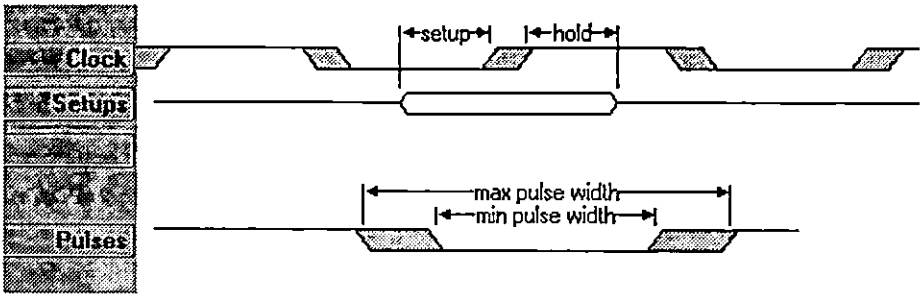
Constraints allow you to automatically check critical parameters that will determine whether your design will work. The topics discussed in this chapter include:

- a description of how constraints function.
- how margins are calculated.
- how to work with constraints—adding, editing, entering numeric values.
- using signal skews.

To get the most use out of the information in this chapter, you'll need to be familiar with a number of terms introduced in Chapter 4, "About TimingDesigner." Turn there as necessary, paying special attention to the sections "Constraints," and "Terminology."

How Constraints Work

A timing constraint establishes a relationship between two edges that must be maintained for the circuit to function. Setup and hold times are examples of constraints. Constraints are established in a timing diagram by indicating source and target edges, and in the Parameter Spreadsheet by entering a Min and a Max. The following illustration shows several types of constraints, a setup and hold, and a pulse width.



The minimum and maximum values for a constraint define tests that determine when the constraint is violated or satisfied. Testing is done by analyzing Measure.

About Measure

When a constraint is analyzed, two measurements can be made: inside and outside.

Inside The measurement is taken from the latest of the source edge to the earliest of the target. The formula for inside measure is *t.e - s.l*. The Min for a constraint determines the inside measure. The min pulse width in the illustration above is an example.

Outside The measurement is taken from the earliest of the source to the latest of the target. The formula for outside measure is *t.l - s.e*. The Max for a constraint determines the outside measure. The max pulse width in the illustration above is an example.

Because these measurements are actually conditions which *must be met*, a constraint that fails either test is called violated, and its representation in the timing diagram and its margin values in the Parameter Spreadsheet turn red (both are green, otherwise, and the the test is satisfied).

Measure can be used as the label of a constraint (instead of its spreadsheet row number, for example). Measure appears between curly braces. A typical measure looks like: {5.0,10.0}.

Constraint Values

Constraint values are Mins and/or Maxs recorded in the Parameter Spreadsheet. If you add a constraint to a timing diagram but don't enter any values for it, no test is performed, and the constraint is ignored.

Most of the time you'll enter a minimum, because the most common problem in a design is not having enough time. A minimum value of 10ns for a constraint means, for example, that the source edge must arrive no earlier than 10ns before the target edge.

Margins

TimingDesigner automatically calculates the margin associated with each constraint. Margin (shown in the Parameter Spreadsheet) indicates the amount by which the constraint is satisfied or violated. Margins appear in the Margin column of the Parameter Spreadsheet between angle brackets. A typical margin looks like this:

<10,15>

The first value, the Min, tells you about the inside dimension; the second value, the Max, tells you about the outside. But margins may also contain negative values. A negative value in a constraint indicates that the constraint is violated. A violated constraint margin value might look like this:

<-10,5>

In this example, the negative value for the Min (the inside dimension) indicates that the width of the measurement is 10ns too large—that the edges are 15ns too far apart in the worst case. This timing deficit must be corrected before your design will function correctly.

Calculating margins

Margin calculations are, as mentioned above, measurements of time available beyond that required by the constraint. The formulas for margins are summarized as:

Inside (minimum):
t.e - s.l - min

Outside (maximum):
max - (t.l - s.e)

About Signal Skews

A signal skew consists of a list of signals and a percentage value, both recorded in the Parameter Spreadsheet. Defining a signal skew modifies the margin so that if the critical path to the source and to the target of the constraint contains one of the signals in the skew, the margin and the measure is increased by:

(1 - k) * (Max - Min)

where *k* is the skew value, *Min* is the minimum, and *Max* is the maximum of the delay involved.

Signal skews are covered in detail later in this chapter.

Working with Constraints

Adding a constraint

✔ To add a constraint

1. Click the Constraint button. The Constraint Source tool appears.
2. Click on the source edge of the constraint. The source edge is selected, and the Constraint Target tool appears.
3. Click on the target edge of the constraint. The target edge is selected, and the system draws the constraint. The Constraint Source tool returns.

Immediately after adding a constraint, you can then do one of the following:

- Add another constraint.
- Change to another tool by clicking a tool button.
- Enter the Min/Max for the constraint in the Parameter Spreadsheet.

Deleting a constraint

✔ To delete a constraint, do one of the following

- In the Diagram Window, select the constraint by selecting its label, and then click the Delete button.
- In the Diagram Window, delete one of the edges of the constraint.

- In the Parameter Spreadsheet, select the row number of the constraint, and then select the Delete command from the Paramater Spreadsheet Edit menu.

Entering numeric values for a constraint

When you add a constraint to a timing diagram, the constraint does not cause any tests to be made until it has numeric values associated with it.

✔ To add numeric values for a constraint

1. Immediately after adding the target edge of the constraint, activate the Parameter Spreadsheet, and locate the row for the constraint.
2. Enter a formula or a Min/Max.

Alternatively:

1. Immediately after adding the target edge of the constraint, click the Pointer button for the Pointer tool.
2. Double-click on the constraint label. The Constraint Attributes dialog box appears.
3. In the Formula column, enter a formula, or a Min/Max (such as [5,10]), and then click OK. The values are copied to the Parameter Spreadsheet.

   Note: The Constraint Attributes dialog box is described in the next section.

Hiding a constraint

✔ To hide a constraint

1. In the Diagram Window, welect the constraint by selecting its label.
2. Select the Hide command from the View menu. The constraint disappears from the timing diagram.

Alternatively:

1. Double-click on the constraint label. The Constraint Attributes dialog box appears.
2. Check Hide, and then click OK. The constraint disappears from the timing diagram.

Alternatively:

1. In the Parameter Spreadsheet, select the constraint by selecting its row number.
2. Select the Hide command from the View menu. The row number in the Parameter Spreadsheet turns light gray to indicate that the constraint in the diagram is hidden.

Showing a hidden constraint

✔ To show a hidden constraint

1. Activate the Parameter Spreadsheet.
2. Select the row of the hidden constraint by clicking on the row number.
3. Select the Show command from the View menu in the Parameter Spreadsheet. When you activate the Diagram Window the constraint will have reappeared.

Alternatively:

1. Select the Display Preferences command on the View menu. The Display Preferences dialog box appears.
2. Click the Gray option in the Hidden field, and then click OK.

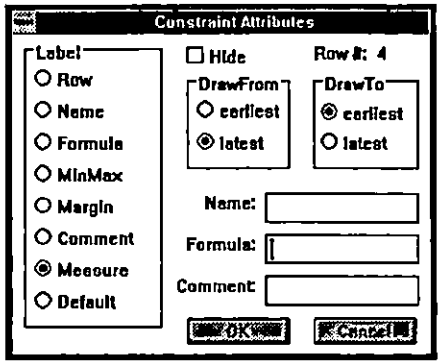
All hidden items are now gray instead of invisible, allowing you to choose the objects to show fully.

Setting and Changing Constraints

You can edit the attributes of a constraint by using the Constraint Attributes dialog box.

✔ To set or change constraint attributes

Double-click on the constraint label; or select the constraint, and then select the Attributes command from the Edit menu. The Constraint Attributes dialog box appears:



Constraint Attributes dialog box fields

Label	These buttons are used to specify the label for the constraint (for more information on labels, see the section "Display Preferences dialog box fields" in Chapter 8, "Delays").
Hide	Checking this box causes the constraint to be hidden.
Row number	Lists the row number of this constraint in the Parameter Spreadsheet. This makes it easy for you to check on the row number for this constraint when the label is something other than Row.
Draw From and Draw To	These fields allow you to set how the constraint is graphically displayed. The Draw From options determine where the source is indicated, and the Draw To options determines the target.
Spreadsheet fields	The Name, Formula, and Comment (comment) fields contain entries for this constraint from the Parameter Spreadsheet. If you edit these fields here, the edits are copied to the Parameter Spreadsheet. The Formula field can be filled with either a formula or a Min/Max.

Labels

Each constraint can have a different label. For information on labels, and on setting the default for constraint labels, see "Setting and Changing Display Preferences," in Chapter 8, "Delays."

Working with Signal Skews

Using signal skews in your designs is a way to take advantage of relationships that you know exist between two specific delays. Manufacturers of parts occasionally publish skew values; the value is a percentage increase in the timing margin that can be claimed because the signals are physically within the same physical part, and are subject to identical physical and manufacturing conditions. Skews are easily implemented by an entry in the Parameter Spreadsheet.

✔ To specify a signal skew

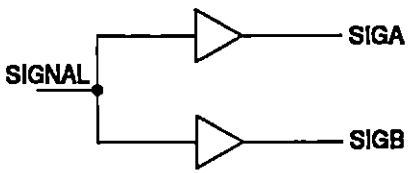
1. Identify the signals involved.
2. In the Parameter Spreadsheet, select the Add Signal Skew command from the Edit menu. A new row appears.
3. In the Name column, enter the signals that are related by the skew, separated by a comma only. For example:

   Sig1,Sig2,DATA

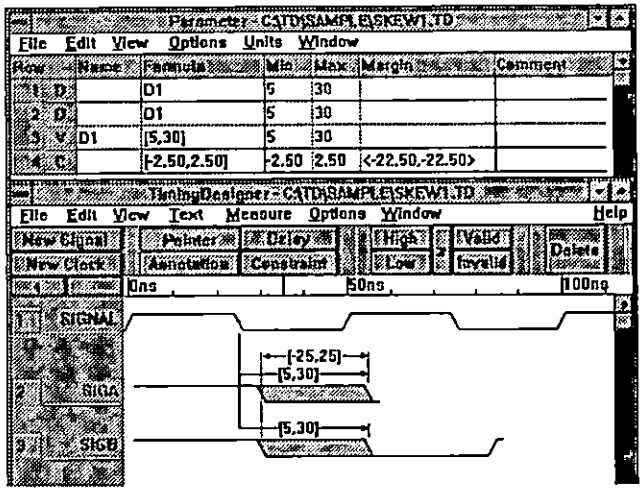
4. In the Formula column, enter the percentage value of the skew.

Skew example

The following example shows how a skew is implemented. Pictured below is a schematic representation of a buffered signal. The buffers introduce a propagation delay of [5,30].



Pictured below is the schematic as represented in TimingDesigner. Both the timing diagram and the Parameter Spreadsheet are shown.



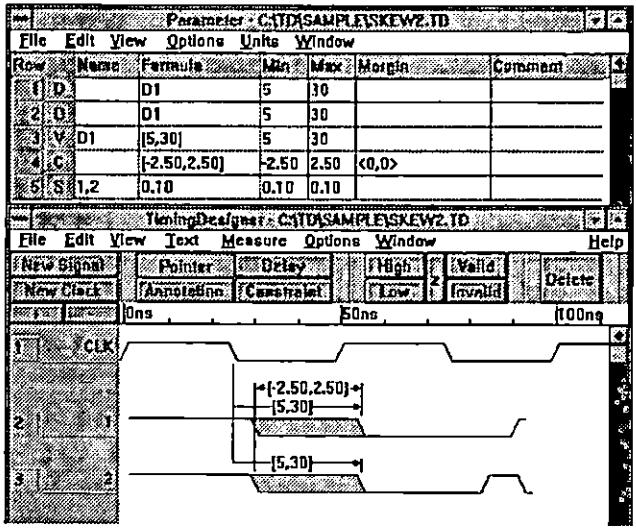
Note the following:

- The delay value of [5,30] is the delay associated with each buffer.
- The value of the constraint shown in Row 4 of the Parameter Spreadsheet establishes the tolerance we expect out of the design. The Min/Max values indicate that the circuit will not work if there is more than 5ns difference between the arrival of SIGA and SIGB.
- The margin shown in the Parameter Spreadsheet reflects the distance between the two edges (25ns, which is also the width of the uncertainty on the signals A and B) minus the Min/Max of the constraint.

As it stands, the design fails because it is out of margin. At this point, let's introduce the signal skew(see the illustration on the next page). Suppose that the gates that generate SIGA and SIGB are both physically in the same package. When this is the case, manufacturers can guarantee better performance. If we introduce a skew of 10% (look at Row 5 in the spreadsheet), the following picture results:

Note the following:

- The margin is now <0,0>.
- The measure of the constraint between signals A and B is now {-2.50,2.50}.



The formula for computing the skew is as follows:

(1 - k) * (Max - Min)

This yields:
(1 - .10) * (30 - 5)
which yields:
(.90) * (25) = 22.50

This value is added to the margin and the measure.

If you want to experiment with skews, create this timing diagram and vary the different timing parameters to observe what happens.

10

Variables, Libraries, and Formulas

This chapter covers a number of topics all having to do with the numerical values involved in creating timing diagrams. It covers:

- how to use variables.
- how to work with Libraries.
- understanding the TimingDesigner formula language, and how to use it.

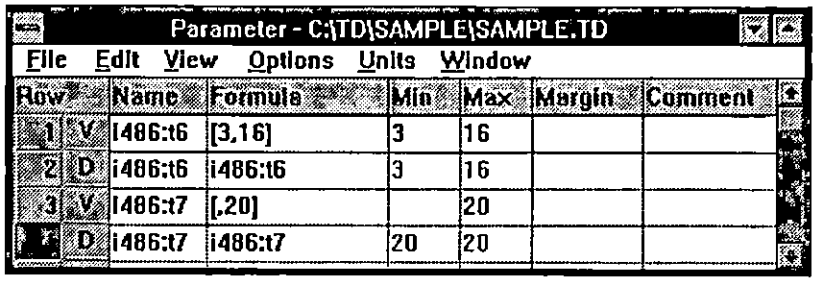
If your use of TimingDesigner is rather simple, the only information you'll need here is on how to enter variables and how to maintain libraries.

If, on the other hand, your work requires the use of complex formulas, you'll also need the information presented on formulas.

Note: For information on controlling the windows, and about spreadsheet column and row descriptions, and special editing techniques, refer to Chapter 5, "TimingDesigner Basics."

Variables

A variable is a spreadsheet row that can be referred to by name in the formulas of other rows. For example, if you have a gate delay of [5,10] that is used in many places in your design, it is easy to create a variable named DEL1 and use it, rather than typing [5,10] in many places.



Row		Name	Formula	Min	Max	Margin	Comment
1	V	i486:t6	[3,16]	3	16		
2	D	i486:t6	i486:t6	3	16		
3	V	i486:t7	[,20]		20		
4	D	i486:t7	i486:t7	20	20		

In the illustration above, the variable in Row 1 is referenced in the Formula column of the delay in Row 2. The same is true for Rows 3 and 4.

Variable names

A variable name is an alphanumeric string. This string, entered into the Name column, is how the variable is referenced. The following rules apply to naming variables:

- Names consist of alphanumeric characters, and are case-sensitive.
- Names may begin with a number, but the number must be followed by one or more alphabetic characters. For example, 1 is not a valid name, but 1x is.
- Names cannot contain spaces.
- Each variable within a spreadsheet must have a unique name. If you enter a duplicate name, TimingDesigner automatically appends a numeral to the name to differentiate it.

Creating a variable

✔ To create a new variable (both spreadsheets)

In either spreadsheet, select the Add Variable command from the Edit menu. A blank variable row is added to the bottom of the spreadsheet. You can now enter information for the variable.

Deleting a variable

✔ To delete a variable (both spreadsheets)

1. Select the variable or variables to be deleted.
2. Select the Delete command from the Edit menu.

   Note: Variables that are referenced in the Formula fields of other spreadsheet rows cannot be deleted.

Moving a variable

✔ To move a variable (both spreadsheets)

1. Select one or more variables to be moved.
2. Select the Move variable command from the Edit menu. The Move tool appears.
3. Position the Move tool on another row, and then click. The variables are moved above the row you indicated.

Using a variable

✔ To use a variable (simple method)

1. Make sure that the variable you want to use is present in the spreadsheet.
2. In the formula column of a delay, constraint, or other variable, enter the name of the variable you want to use. When you press Enter or click outside the field, the values of the row will be updated with the variable information.

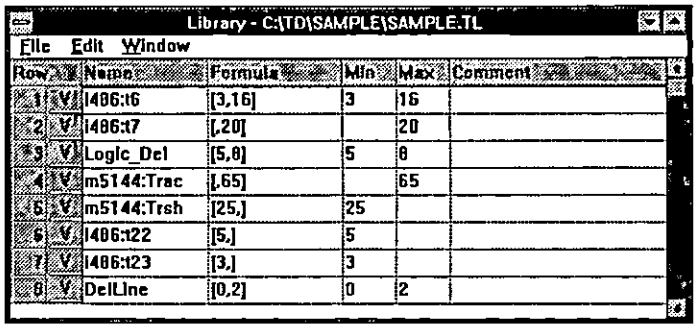
   Note: If the variable you enter is combined with other elements to create an formula, the Min/Max values will be gray, indicating that they are derived from the formula. You can not directly edit gray min/max values.

   Note: Circular references, such as referencing a variable in its own formula field, are not allowed.

Other ways of using variables are presented in the next section.

Using Libraries

Libraries are used to store lists of variables that are commonly used in your design. By creating and maintaining these lists of variables, you can greatly speed your work and reduce errors. A typical library file looks like this:



Row	V	Name	Formula	Min	Max	Comment
1	V	i486:t6	[3,16]	3	16	
2	V	i486:t7	[,20]		20	
3	V	Logic_Del	[5,8]	5	8	
4	V	m5144:Trac	[,65]		65	
5	V	m5144:Trsh	[25,]	25		
6	V	i486:t22	[5,]	5		
7	V	i486:t23	[3,]	3		
8	V	DelLine	[0,2]	0	2	

Notice that libraries contain only variables. You may create as many libraries as you need, but only one can be open at a time during a TimingDesigner session.

Using library variables (importing)

Variables stored in the Library Spreadsheet can be brought into a Parameter Spreadsheet in one of two ways. As you work, you'll probably use a combination of the two, depending upon what you're doing.

Sending variables

A variable or group of variables can be sent from one spreadsheet to another. For instance, if a certain design demands a new set of variables and you enter them in the Parameter Spreadsheet for the design, you can then send them as a batch to a Library.

✔ To send variables

1. In the source window, select one or more variables to be sent. For instance, if you want to send a variable from the Library sent to the Parameter Spreadsheet, then select the variable in the Library Spreadsheet.
2. Select the Send To command from the Edit menu in the source window. The selected variables are copied to the destination spreadsheet.

Importing by reference

When you reference a variable that does not exist in the Parameter Spreadsheet, TimingDesigner imports the variable for you from the Library.

✔ To import a variable by reference

1. Make sure that a Library is open, and make sure that the variable you are going to reference is in the Library.
2. In the formula column of the Parameter Spreadsheet, enter the variable name. It can be anywhere in the column, even as a part of a more complex formula.
3. When you press Enter or leave the field, TimingDesigner evaluates the formula. Part of the process is to import the referenced variable. The imported variable appears at the bottom of the spreadsheet.

   Note: Circular references, such as referencing a variable in its own Formula field, are deleted and disallowed.

Updating variables

Occasionally you'll change a variable definition in one spreadsheet or the other. TimingDesigner allows you to update the other spreadsheet. This function exists because once you have a variable of a given name in a spreadsheet, you cannot copy the variable.

✔ To update variables

When it is time to update variables, select the Update from *Spreadsheet Name* command on the Edit menu in the destination spreadsheet. For example, if you've changed variables in a Parameter Spreadsheet, you would select the Update from Parameter Sheet command on the Edit menu in the Library Spreadsheet. The variables are updated.

Library File Format

TimingDesigner Library data is stored as a sequence of records where each line has five fields separated by commas. This format is directly compatible with the CSV (Comma Separated Value) format in popular spreadsheets, such as Microsoft's Excel.

1. Name	The name of the variable. Spaces are not allowed, and may not start with a number.
2. Formula	If an formula is present, both the Minimum and Maximum fields are ignored.

3. Minimum	Must be a numeric entry. If omitted, the minimum takes on a null value.
4. Maximum	Must be a numeric entry. If omitted, the maximum takes on a null value.
5. Comment	A free-form alphanumeric field in quotes (" ").

Omitting fields

To omit a field, use two commas (,,). Most often you will omit the Formula field, and entries will look similar to this:

```
tphl,,5,10,"Propagation delay from high to low"
```

This usage corresponds to the common situation in which library data is a direct transcription of information contained in vendor component data books.

Commas in fields

Any field that contains a comma must be enclosed by double-quotes (" ").

Managing Library hiles

The File commands in the Library window are used to manage Library files.

New	Opens a new, blank Library file. If the current Library Spreadsheet has unsaved data in it, you are given the opportunity to save it.
Open	Opens an existing Library file.
Save	Saves the current Library file. If the contents of the Library file have never been saved to a file, the File dialog box appears, where you can assign a file name.
Save as	Saves the current Library file to a new name.

See Chapter 5, "TimingDesigner Basics," for information on the File dialog box.

See Chapter 12, "Printing and Exporting," for information on printing spreadsheets.

Formulas

All mathematical formulas in TimingDesigner evaluate to the form *[x,y]*, where *x* is a minimum and y is a maximum. The reason is that formulas reference propagation delays and setup and hold times that are specified as minimums and maximums. A value in the form of *[x,y]* is often referred to as a Min/Max.

The purpose of formulas in TimingDesigner is to allow you to define variables, and then use those variables in combination with numbers, operators, and other variables. For instance, you can create a variable called *GATE-DELAY* and give it a value of [5,10] (minimum and maximum values). You can then use the variable in combination with numbers and operators to derive:

```
2*GATE_DELAY
```

Or, you could combine it with another variable:

```
GATE_DELAY+TIME1
```

Formula elements

The elements used to form formulas are simple: words, numbers, operators, and other sub-formulas.

Number	A number is a sequence of digits; for example, 245. Note that TimingDesigner understands an empty numeric field as a null value, as opposed to a numeric zero. (Nulls are discussed later in this section.)
Min/Max	A Min/Max ([5,10], for example) is a valid element in an formula.
Word (variable names)	A word is a sequence of letters and digits, for example, 245X. Words are used as variable names. Variable names are used in a formula to reference the values of the named variable. *74LS:245:tpd* is an example of a variable name.
Operator	An operator is: - (a unary minus) * (multiplication) / (division) + (addition) - (subtraction)
Sub-formula	Any formula can be used as a sub-formula.

Mathematical operations

Operations in formulas

Whenever mathematical operators affect a formula, minimums combine with minimums, and maximums with maximums.

For instance, the formula *2*GATE_DELAY*, where GATE_DELAY is equal to [10,15], would be evaluated as follows (spaces in the formulas below are for clarity):

```
  [10,15]
* [ 2, 2]
= [20,30]
```

Two variables in combination with an operator behave in the same way. For instance, *GATE_DELAY+T_DELAY*, where *GATE_DELAY* equals [10,15] and *T_DELAY* equals [2,5.5], would be evaluated as follows (spaces in the formulas below are for clarity):

```
  [10,15.0]
+ [ 2, 5.5]
= [12,17.5]
```

Operation order

Mathematical operations in formulas are performed left-to-right and, unless altered by parentheses, in this order:

first:	- (unary minus)
second:	* (multiplication) and / (division)
third:	+ (addition) and - (subtraction)

For instance, *1+2*3+4* is operated on in this order:

```
1+(2*3)+4
1+(6)+4
(7)+4
11
```

Parentheses can be used to alter the default order of operation by indicating which operations to perform first. If the equation above were modified to be *(1+2)*(3+4)*, it would be operated on in this order:

```
(1+2)*(3+4)
(3)*(7)
21
```

Functions

The formula language has two functions:

min(variable)
max(variable)

These functions allow you to recombine min and max values of different variables, or with other values. These functions are always contained within brackets, for example:

[min(variable), min(variable)]
[min(variable), max(variable)]
[max(variable), min(variable)]
[max(varaible), max(variable)]

For instance, assume that you have two variables, VAR1 whose min/max values are [9,15], and VAR2 with min/max values of [5,25]. Further assume that you need to use the min value of one variable and the max of the other variable as a constraint. You can use the formula:

[min(VAR1),max(VAR2)]

which results in a min/max value of [9,25] for the constraint.

Case is important when using this function:

- The operators *min* and *max* must be lower case.
- The case of the variable names used in the formula must match the case of the actual variable names.

About Nulls

The meaning of null values in a Min/Max depends upon whether the Min/Max is used as a delay or a constraint. The following table summarizes how nulls are interpreted.

Delays	[,] = delay does not affect the timing of its associated edges [5,] = [5,5] (nulls automatically supressed in final evaluation) [,7] = [7,7] (nulls automatically supressed in final evaluation)
Constraints	[,] = constraint always satisfied (green) [5,] = constraint satisfied if minimum measurement is >/= 5 [,7] = constraint satisfied if maximum measurement is </= 7 [5,7] = constraint satisfied if minimum measurement >=5 and maximum measurement <=7

Null values propagate as identity values. Performing any operation between a null and a number results in the number. For example:

[5,]*[5,30] results in [25,30]
[5,]+[5,20] results in [10,20]
[5,]/[5,30] results in [1,30]
[5,]-[5,30] results in [0,30]

11

Text Annotations

Text annotations are comments placed at certain locations on the timing diagram. The topics discussed in this chapter cover:

- how to add and edit text annotations
- how to work with text attachments
- how to control the font used for annotations

Working with Annotations

Each annotation is contained within a bounding box, and the text within the box is called an annotation. Annotations can be moved anywhere within a design. Other important things to know about text:

- a font attribute can be specified for each annotation.
- all keyboard characters are valid within annotations, including Tab.
- standard Microsoft Windows editing procedures are available within annotations.
- the Delete button (in the Tool Bar) is used to delete an entire annotation. *Do not use it to delete words or characters within an annotation!* Use the Delete and Backspace keys. If you click the Delete tool while you are editing an annotation, the entire annotation will be deleted.

Adding annotations

Annotations can be added anywhere in a drawing. Unlike delays and constraints, annotations cannot be hidden, so once you add an annotation you may have to move it if it overlaps other objects. When you add an annotation, it is attached to the edge closest to where the annotation begins.

See the section, "About SmartSnap," later in this chapter, for information about automatic alignment of annotations within waveforms.

See the section, "Attaching annotations," later in this chapter, for information about attachments.

✔ To add an annotation

1. Click Annotation button. The Crosshair tool appears.
2. Click where you want the annotation to begin. The insertion point appears, enclosed by a small bounding box.
3. Begin typing. The bounding box expands to accomodate the text. If you are entering more than a few words and need line breaks, press Enter.

While working in an annotation you can:

- Edit the text using standard Windows editing techniques. The pointer is an I-beam tool inside of annotations.
- Change the font (see the section "Setting the Text Font," later in this chapter).

When you're done with the annotation, you can:

- Move the pointer (the Crosshair tool) to a new location, and then click to create a new annotation.
- Select a menu command or another tool.

Editing annotations

✔ To edit an annotation

1. Select the Annotation tool.
2. Select the annotation to edit. The insertion point appears inside of the annotation. Edit as required.
3. When finished, select another annotation, or any menu command or tool.

   Note: To delete characters within an annotation use the Backspace and Delete keys on the keyboard. The Delete tool in the tool bar deletes the entire annotation.

Moving annotations

✔ To move an annotation

Using the Pointer tool drag the annotation to a new location.

Note: If SmartSnap is on, the annotation exhibits certain behaviors. See the section, "Attaching Annotations," later in this chapter, for more information.

Deleting annotations

✔ To delete an annotation

1. Select the annotation using the Pointer tool.
2. Click the Delete button.

Attaching Annotations

Annotations are automatically attached to the closest edge when they are created. The attachment is indicated by a "string" that extends from the annotation to an edge that is visible when the annotation is selected. An annotation attached to an edge moves along with the edge. This is especially useful for annotations that must be physically close to a certain event in a circuit.

Note that all annotations are attached to some edge. If an annotation is not attached to an edge somewhere in a waveform, it is attached to the implied edge at time 0.

✔ To detach an annotation from an edge

1. Select the annotation.
2. Select the Detach from Edge command on the Text menu. The string snaps to time 0.

✔ To choose a new attachment

1. Select the annotation.
2. Select the Attach to Edge command on the Text menu. The string is now attached to the Crosshair tool.
3. Click on the new attachment edge. If SmartSnap is on, the annotation automatically aligns itself with the waveform.

About SmartSnap

The SmartSnap feature automatically determines alignment for annotations. SmartSnap tries to place annotations in the horizontal plane of the signal to which they are associated. If this isn't possible, text is placed directly above the attachment point. SmartSnap also changes the way attachments are handled; when SmartSnap is on, the attachment jumps from edge to edge as you move an annotion around the diagram.

SmartSnap can be used in two ways:

- **While adding annotations**—All new annotations are automatically aligned with the signal to which they are attached.
- **After adding annotations**—In this case, each time you select an annotation with the Pointer tool, the annotation will become aligned. In addition, if you move an annotation, the "string" will snap from edge to edge, allowing you to see the attachment point.

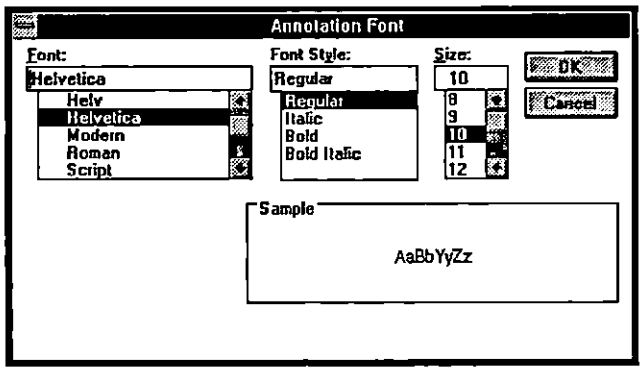
The best way to understand how SmartSnap works is to experiment with it.

✔ To experiment with SmartSnap

1. In a drawing with three or four waveforms with multiple edges, enable SmartSnap.
2. Select the Text tool, and then create an annotation.
3. Select the Pointer tool, and then select the annotation you just created.
4. Drag the annotation around the diagram. You can see how the "string" jumps from edge to edge. Release the left mouse button to place the annotation.
5. Repeat this a few more time, experimenting with different placements. Notice how SmartSnap determines right- or left-justification of the text in relation to the attachment point.

Setting the Annotation Font

Each annotation can be assigned one set of text attributes that affect the appearance of the characters in the annotation. This is done using the Font dialog box.



Annotation Font dialog box fields

The fields in the Annotation Font dialog box allow you to do the following:

Font Determines the typeface used for all text in the annotation. Available type faces are determined by your Microsoft Windows installation.

Font style	Allows you to specify whether all text in the annotation should be italic, bold, or bold italic.
Size	Determines the size in points for all text in the annotation.
Example box	Allows you see an example of the font specified.

✔ To set the default font

Note: To change the default without affecting an existing annotation, make sure no annotation is selected before performing these steps. If you perform these steps with an annotation selected, the new defaults will be assigned to the selected annotation as well.

1. Select the Annotation Font command from the Text menu. The Annotation Font dialog box appears.
2. Make the desired selections, and check the Make Default check box.
3. Click OK when done. The attributes you set will be in force for all subsequent annotations.

✔ To set text attributes for an annotation

1. Select an annotation.
2. Select the Annotation Font command from the Text menu. The Annotation Font dialog box appears.
3. Make the desired selections, and then click OK.

12

Printing and Exporting

Printing

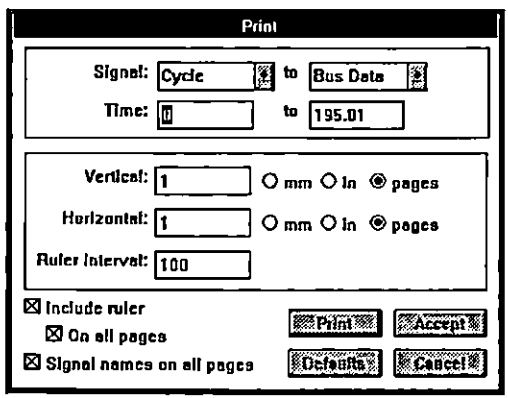
You can print from all TimingDesigner windows.

Printing timing diagrams

You can print timing diagrams, Parameter Spreadsheets, or Library Spreadsheets. The process for each is similar, and is done by using the Print dialog box.

✔ To print a timing diagram

1. Choose a printer, if necesary (see section following).
2. Select the Print command from the File menu. The Print dialog box appears.



3. Make the appropriate selections and click OK when done. For information about the choices in the dialog box, see the following section, "About the Print dialog box."

About the Print dialog box (Diagram Window)

The illustration below shows the Print dialog box as it would appear if selected from the Diagram Window. The dialog box has three main areas.

What to print

The Signals drop-down boxes specify the range of signals to be printed. The Time entry fields specify the length of the time line to be printed.

How to print

- The Vertical field and options buttons specify the vertical space allotted to the the chosen data. For example, if you choose the Pages option, all data is printed on that number of pages vertically.
- The Horizontal fields and option buttons specify the horizontal space allotted to the chosen data.
- The Ruler interval field specifies the interval between major ticks in the ruler (major ticks are those with numbers). Four minor ticks will separate major ticks regardless of the interval chosen. For example, if you enter 100, the ruler will have ticks and numbers every 100ns, and ticks at 25ns intervals.

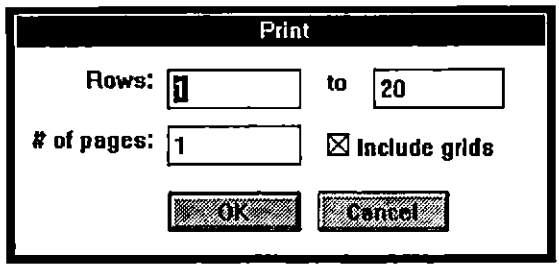
Commands and options

- Enable the Include ruler option if you want the ruler to print on top pages. Enable the On all pages option if you want the ruler to appear on all pages when output is tiled.
- Enable the Signal names on all pages option when you want signal names to appear on each page. Otherwise they appear only on the left-most page when output is tiled.
- Click the Print button to begin printing
- Click the Accept button to save the current settings of the How to print fields for the current session only.
- Click the Defaults button to return the dialog box to its original settings.
- Click the Cancel button to exit the dialog box without taking any action.

Printing spreadsheets

✔ To print a spreadsheet

1. Choose a printer, if necesary (see section following).
2. Select the Print command from the File menu of the Parameter Spreadsheet or the Library Spreadsheet. The Print dialog box appears.



3. Specify the print parameters and click OK.

About the Print dialog box (spreadsheet windows)

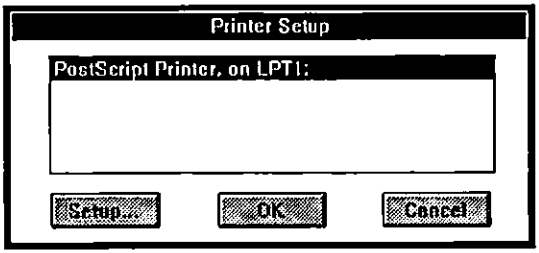
- Use the Rows fields to specify the number of rows to print.
- Use the of pages field to specify how many pages to use when printing.
- Enable the Include grids option when you want the horizontal and vertical lines printed.

Choosing a printer

Occasionally you may need to choose another printer or change its setup. Follow the steps below to do this.

✔ To set or change your printer

1. Select the Printer Setup command. The Printer Setup dialog box appears (see below).
2. Select the printer you want to use and click OK.
3. If necessary, click Setup to verify or change your printer settings. Refer to your Microsoft Windows documentation for information on the Setup dialog box.



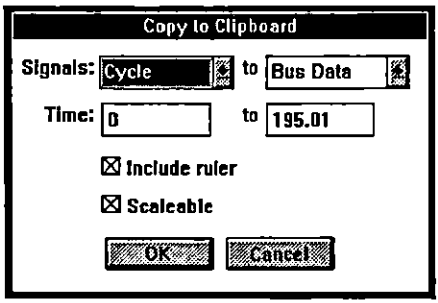
Exporting Graphics

TimingDesigner allows you to export graphic files of the timing diagrams or the spreadsheets. The process for each is similar, and is done by using the Copy to Clipboard dialog box. The information transferred to the clipboard is in Windows Metafile format. The results of pasting the data into another application, such as a word processor, depend largely upon how the applications handles the Metafile format.

Exporting timing diagrams

✔ To export a timing diagram

1. Select the Copy to Clipboard command from the Edit menu in the Diagram Window. The Copy to Clipboard dialog box appears.



2. Make the appropriate selections and click OK when done.

Copy to Clipboard dialog box (Diagram Window) fields

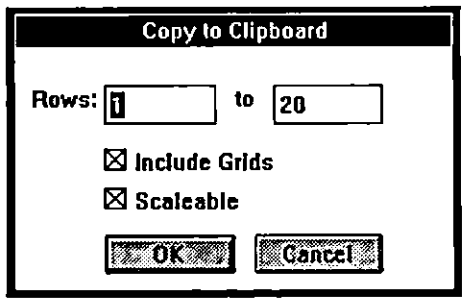
Signals	Specifies the range of waveforms to be copied.
Time	Specifies the length of time line to be copied.

Include ruler	Causes the ruler to appear.
Scalable	Used when you want the image to be scalable in the destination application. Some applications import scalable images better. Some import non-scalable images better. You will need to find the combination that is best for your application.

Exporting spreadsheets

✔ To export a spreadsheet

1. Select the Copy to Clipboard command from the Edit menu in the Parameter Spreadsheet or Library Spreadsheet. The Copy to Clipboard dialog box appears.



2. Make the appropriate selections and click OK when done.

Copy to Clipboard dialog box (spreadsheets) fields

Rows	Specifies the range of rows to be copied.
Include grids	Causes the grids to appear.
Scalable	Used when you want the image to be scalable in the destination application. Some applications import scalable images better. Some import non-scalable images better. You will need to find the combination that is best for your application.

13

Other Features and Functions

This chapter contains information on a number of TimingDesigner features:

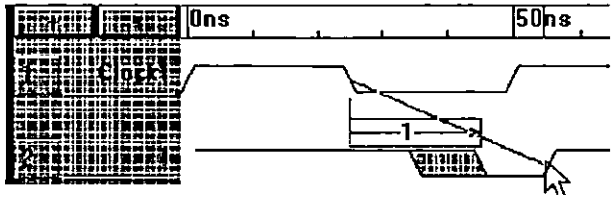
- changing the source or target of a delay or constraint
- taking time measurements (using the vertical reference cursor and marker line)
- moving around timing diagrams using the Go To command
- using horizontal reference guides
- setting the font for timing diagrams and spreadsheets
- merging timing diagrams
- how to save TimingDesigner options

Changing Sources and Targets

You can change the source or target of a delay or constraint.

✔ To change a source or target

1. Select a delay or constraint.
2. Select either the Choose New Source or Choose New Target command from the Edit menu. A "string" stretches from the stationary edge to the pointer, allowing you to keep track of the source or target.



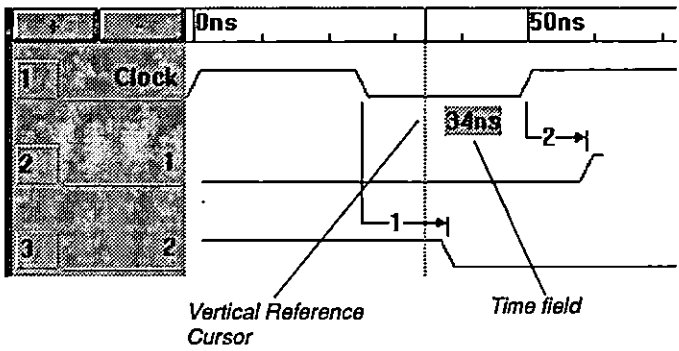
In the example above, the target edge is being moved.

3. Click on the new source or target edge. The delay or constraint is redrawn and the Pointer tool reappears.

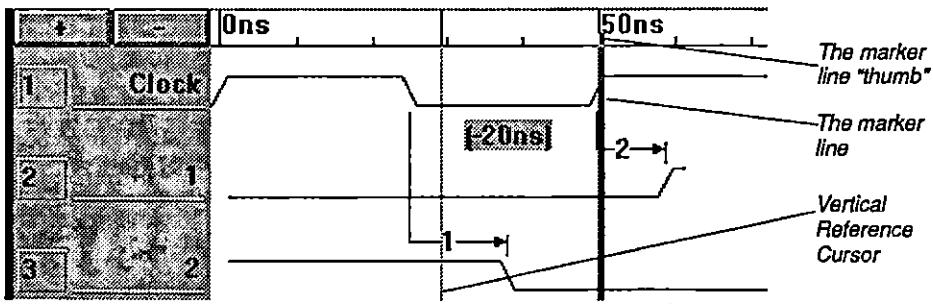
Taking Time Measurements

Two features help you take time measurements: the vertical reference cursor and the marker line.

The reference cursor is a thin vertical line that extends through the timing diagram. The line is attached to the current pointer (Pointer, Text, Delay, or Constraint tool), and is coupled with a numerical field, called the time field. As you move the pointer around the timing diagram, the time field indicates the location in time of the pointer (relative to 0), and the vertical reference cursor allows you to carefully check how edges are lined up.



The marker line is a vertical bar that you drop in a location. Once the marker line is established, the value in the time field is relative to the marker, rather than time 0. This allows you to take accurate relative measurements. Relative measurements are indicated in the time field by putting the time value in parentheses.



Working with the vertical reference cursor

✔ To toggle the vertical reference cursor

Select the Reference Cursor command from the Measure menu.

Working with the marker line

The vertical reference cursor must be toggled in order for the marker line to work. You can have only one marker line per drawing.

Setting the marker line location

You can set the location of the marker line in two ways: in any location, or related to an edge. The method you use depends upon the type of measurement you want to take.

✔ To set the marker line anywhere

Click in the ruler. The marker line appears. Notice that the time field now measures relative to the marker line, and this is indicated by the time value appearing in parentheses. Negative values are indicated by a minus sign prefix.

✔ To set an edge-related marker line

1. Select an edge.
2. Select the Measure from Earliest command or the Measure from Latest command from the Measure menu. The marker line appears running through the indicated edge.

Moving and removing the marker line

✔ To move the marker line

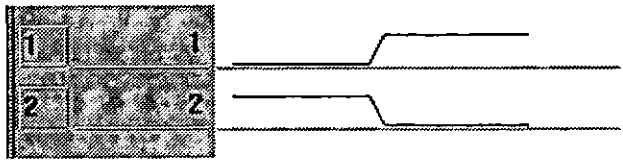
Click in the ruler in another location. The old marker line disappears, and a new one appears at the cursor location.

✔ To remove the marker line

Click on the marker line thumb, the small black portion inside of the ruler. The marker line disappears. Alternatively, select the Measure from Zero command from the Measure menu to remove the marker line from the diagram.

Horizontal Reference Lines

Depending upon the type of work you do, you may find it convenient to use the horizontal reference lines when adding edges to your diagrams. These light gray lines lines appear at the bottom of each waveform, and allow you to get a better idea of where to click to place edges.



✔ To toggle horizontal reference lines

Select the Horizontal Reference Lines command from the Options menu.

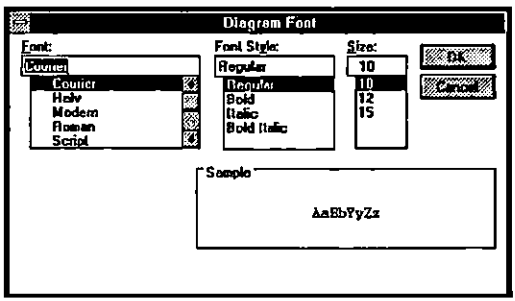
Setting the Diagram and Spreadsheet Font

The font of the text elements in the Diagram Window and in the Spreadsheets can be changed. You can change these fonts as needed.

In the Diagram Window, the changeable elements are the ruler, the waveform names and numbers, and the delay and constraint labels. In the Spreadsheets, the changeable elements are the column headings and the contents of the cells.

✔ To set the font for timing diagrams

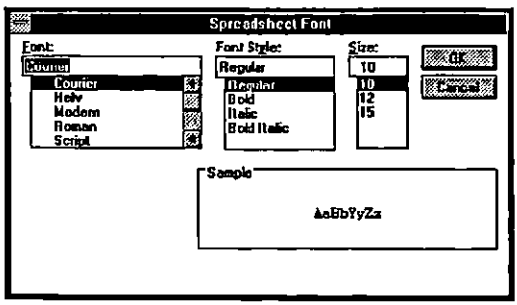
1. Select the Diagram Font command from the Options menu in the Diagram Window. The Diagram Font dialog box appears.



2. Make the appropriate selections and click OK. The changes take effect immediately.

✔ To set the font for the spreadsheets

1. Select the Spreadsheet Font command from the Options menu in the Parameter Spreadsheet. The Spreadsheet Font dialog box appears.



2. Make the approriate selections and click OK. The changes take effect immediately.

Note: Changing the font for one spreadsheet changes both of the spreadsheet fonts.

Saving TimingDesigner Options

The way you have configured certain TimingDesigner options can be saved. When you save these options, TimingDesigner will be in a familiar state the next time you start it. The options that can be saved are:

- Horizontal Reference Lines setting
- Vertical Reference Cursor setting
- Diagram font
- Spreadsheet font
- Skip Signal Dialog setting
- Spreadsheet column positions

Options are saved in a file called TD.INI in your Windows directory.

✔ To save TimingDesigner options

1. Configure TimingDesigner options to your liking.
2. Before exiting TimingDesigner, enable the Save Settings On Exit command on the Options menu.

✔ To change TimingDesigner options

When the Save Settings On Exit command is toggled on, the current settings are preserved.

✔ To reset TimingDesigner options to default

Delete the file td.ini. When you start TimingDesigner it will be in its default state. If you don't want the current options saved, remember to toggle off the Save Settings On Exit command on the Options menu.

TD.INI File Format

The file format used for TD.INI is similar to all Window .INI files. The example shows the contents of a TD.INI file.

```
[Timing Designer Configuration]
DiagramFont=System 10
ParamFont=System 10
ParamColumns=0 8 16 31 37 43 60
LibColumns=0 8 16 31 37 43 43
VRefCursor=no
HRefLines=no
SkipSignalDialog=no
SaveOnExit=yes
```

The option names and their related menu commands or functions are:

DiagramFont	The font used in the Diagram Window (Diagram Window, Options menu).
ParamFont	The font used in the Parameter Spreadsheet (Parameter Window, Options menu).
ParamColumns	Column widths in the Parameter Spreadsheet.
LibColumns	Column widths in the Library Spreadsheets.
VRefCursor	Vertical reference cursor (Diagram Window, Measure menu).
HRefLines	Horizontal Reference Guides (Diagram Window, Options menu).
SkipSignalDialog	The Skip Signal Dialog command (Diagram Window, Options menu).
SaveOnExit	The Save On Exit command (Diagram Window, Options menu).

Merging Timing Diagrams

You can merge (or combine) two timing diagrams by using the merge feature. The merge feature imports all waveforms and all data in the Parameter Spreadsheet of another design into an existing design. TimingDiagrams merge according to the following rules:

- The merged waveforms are placed at the bottom of the timing diagram.
- Signal and variable name collisions are automatically handled by TimingDesigner. The signal and variable names of the imported signals are preserved, and the existing duplicate signal and variable names have a number appended to the signal or variable name (the original signal DATA becomes DATA1). If a signal or variable name ends in a number, the number is incremented (the original variable GATE_DEL1 becomes GATE_DEL2).

A timing diagram can be merged with itself.

✔ To merge two timing diagrams

1. Open a timing diagram.
2. Select the Merge command from the File menu in the Diagram Window. The Merge Diagram dialog box appears (it is identical to the Open Diagram dialog box).
3. Select the diagram to be merged and click OK.

Zoom

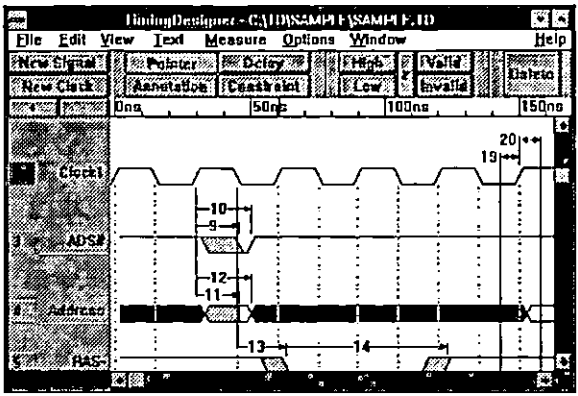
The zoom buttons allow you to adjust the time scale to see more or less of your design.

The + button increases display magnification, showing you greater detail and less time.

The - button decreases display magnification, showing less detail and more time.

Grids

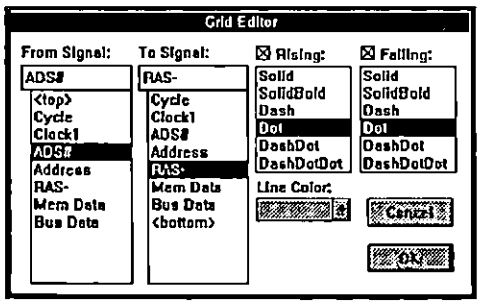
You are able to specify multiple grids in your timing diagram. Displaying grids allows you to see how the signals in your design line up with a given clock.



- Grids are associated with clock edges.
- Each clock signal in a design can have a grid style associated with it.
- A grid line can be generated on all rising and/or all falling edges.
- Each grid line specification can use one of 16 colors and one of 6 line types.

✔ To specify a grid

1. Select a clock waveform.
2. Select the Grid command from the View menu. The Grid Editor dialog box appears (see description below). Specify the parameters of the grid.



Grid dialog box fields

The Grid dialog box is used to specify the grid for a clock signal. The dialog box is shown below, and its fields are explained.

From and To Signal

These data boxes list all of the signals in your design in their physical order. The entry field at the top of each data box allows you to specify how far the grid lines extend. Note that while grid lines are contiguous, they don't have to start at the clock with which they are associated.

To specify a signal, click on the signal name in the data box, or enter the signal name in the entry field.

Rising and Falling Edge/ Line Types

The Rising and Falling edge check boxes are used to specify the clock edge(s) through which the grid lines appear. The data boxes below the check boxes list the available line types.

Line color

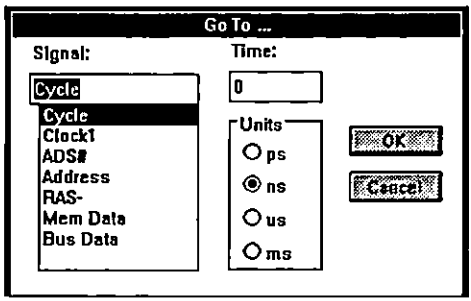
The Line Color combo box sets the line color for both rising and falling grid lines for this clock signal.

Jumping to a Diagram Location

The Go To command is a powerful tool in large timing diagrams. It lets you jump to a given location both in time and space. Using the Go To command you are able to specify both the signal to be displayed and the time at which to view the signal.

✔ To jump to a diagram location

1. Select the Go To command from the Options menu. The Go To dialog box appears.



2. Specify the signal to jump to by clicking on the name in the Siganl data box, or by entering the name in the entry field. The signal you indicate will be placed at the top of the diagram window. The selections First and Last refer respectively to the top-most and bottom-most waveforms.
3. Specify the time to jump to by entering the value in the Time entry field, and indicate the unit of measure for the time entered by clicking the appropriate radio button. Your view of the timing diagram will adjust so that the time you indicate is against the left edge of the timing diagram (next to the signal names).
4. Click OK.

Chronology Corporation

Worldwide Headquarters:
2721 152nd Avenue N.E.
Redmond, WA 98052-5516
(800) 800-6494
(206) 869-4227
(206) 869-4229 fax

To order TimingDesigner, contact your local sales representative or call Chronology Corporation